# WRIA-1 DSS

# USER MANUAL

# Table of Contents

**Installing the WRIA-1 DSS ..... 1**
Starting the Installation ..... 1
.NET Framework Installation ..... 2
Installation Welcome ..... 4
Setup Type ..... 5
Installing ..... 6
InstallShield Portion Complete ..... 7
Microsoft SQL Server Desktop Edition Installation ..... 7
Reboot ..... 8
Restoring Databases ..... 9
Installation Complete ..... 9

**MapWindow Watershed Information System ..... 10**
Overview of the MapWindow WIS Interface ..... 11
Opening the MapWindow WIS ..... 11
Parts of the MapWindow Interface ..... 12
Map ..... 12
Legend ..... 12
Legend Editor ..... 13
Preview Map ..... 15
Select Feature Tool ..... 15
Zoom Tools ..... 16
Layer Tools ..... 17
Customizing the MapWindow WIS Interface ..... 18
Feature Identifier and Shape Labeler Plug-in ..... 19
Feature Identifier ..... 19
Shape Labeler ..... 22
Attribute Table Editor Plug-in ..... 23

**Scenario Builder ..... 25**
Starting the DSS ..... 25
Creating a New Scenario ..... 26
Adding Elements ..... 27
Connecting Elements ..... 28
Viewing Properties and the Scenario Tab ..... 29
Configuring Elements ..... 30
Setting the Date Range ..... 31
Saving the Scenario ..... 31
Running the Scenario ..... 32
Errors ..... 33
Scenario Run Log ..... 33
Importing and Exporting Scenarios ..... 35
Editing the Base Data ..... 36

Configuring the DSS Database ..... 37
Placing a Scenario Inside of another Scenario ..... 37
The Loop Control Node ..... 38
Custom Scenario Elements ..... 41

**Land Cover Change Scenario Element ..... 42**
Add Land Cover Change Element to DSS Scenario ..... 42
Add Change Land Cover to DSS Scenario ..... 42
Add Land Cover Summarizer to DSS Scenario ..... 44
Edit Land Cover Change Scenario Element Settings ..... 46
Select Input Land Cover Grid ..... 47
Select Polygon Shapefile ..... 48
Select Output Land Cover Grid Directory ..... 50
Select Land Cover Change Polygons ..... 50
Edit Land Cover Summarizer Settings ..... 56
Run Land Cover Change Scenario Element ..... 60

**Water Quantity Model Interface ..... 61**
Add Water Quantity Model to DSS Scenario ..... 61
Edit Water Quantity Model Settings ..... 64
Edit Water Quantity Model Inputs ..... 65
Edit Water Quantity Model Outputs ..... 81
Run Water Quantity Model ..... 84

**Lake Whatcom Water Quality Model Interface ..... 85**
Add Lake Whatcom Water Quality Model to DSS Scenario ..... 85
Edit Lake Whatcom Water Quality Model Settings ..... 88
Edit Lake Whatcom Water Quality Model Inputs ..... 88
Edit Lake Whatcom Water Quality Model Outputs ..... 99
Run Lake Whatcom Water Quality Model ..... 100

**WRIA-Wide Water Quality Model Interface ..... 102**
Add WRIA-Wide Water Quality Model to DSS Scenario ..... 102
Edit WRIA-Wide Water Quality Model Settings ..... 105
Edit WRIA-Wide Water Quality Model Inputs ..... 105
Edit WRIA-Wide Water Quality Model Outputs ..... 113
Run WRIA-Wide Water Quality Model ..... 114

**Periodicity Viewer Plug-in ..... 116**
Configuring the Data Connection ..... 116
Selecting the Model Output Nodes Point Shapefile ..... 117
Selecting the Periodicity Database ..... 118
Viewing Periodicity Data ..... 120
Selecting Photo Viewer Points from the Map ..... 120
Viewing Periodicity Data via the Periodicity Viewer Toolbar Button ..... 121
Periodicity Viewer/Editor Options ..... 121

Node Selection ..... 122
Viewing Distribution and Periodicity Data ..... 122
Editing Distribution Data ..... 123
Editing Periodicity Data ..... 123
Restoring Database Values ..... 124
Copy/Save the Distribution and Periodicity Grid ..... 125

**Habitat Time Series Model Interface ..... 126**
Add Habitat Time Series Model to DSS Scenario ..... 126
Edit Habitat Time Series Model Settings ..... 130
Edit Habitat Time Series Model Inputs ..... 130
Edit Habitat Time Series Model Outputs ..... 139
Run Habitat Time Series Model ..... 142
Exporting Scenario Run Log ..... 144

**Temperature and Dissolved Oxygen Flags Model Interface ..... 145**
Add Temperature and DO Flags Model to DSS Scenario ..... 145
Edit Temperature and DO Flags Model Settings ..... 148
Edit Temperature and DO Flags Model Inputs ..... 149
Edit Temperature and DO Flags Model Outputs ..... 155
Run Temperature and DO Flags Model ..... 160
Exporting Scenario Run Log ..... 161

**Time Series Data Analyst Plug-in ..... 163**
Viewing Water Quality Data ..... 163
Viewing Data via the Time Series Data Analyst Toolbar Button ..... 164
Selecting Water Quality Stations from the Map ..... 164
Time Series Analyst Options ..... 165
Plot Window ..... 166
Plot Saving, Exporting, and Printing ..... 167
Plot Types Tabs ..... 169
Selecting Stations and Water Quality Parameters ..... 171
Restrict Data Range ..... 171
Statistical Summary, Data Sources, Plot Options, Plot Enhancements and Notes ..... 171
Correlation Analysis ..... 183
Correlation Analysis ..... 184
Multiple Time Series Analysis ..... 185
Multiple Time Series Analysis ..... 186
Parameter Summary Analysis ..... 189
Statistical Summaries ..... 192
Evaluate Criteria ..... 195
Station Grouping ..... 197

**Photo Viewer Plug-in ..... 200**
Viewing Photographs ..... 201
Selecting Photo Viewer Points from the Map ..... 201

Viewing Photographs via the Photo Viewer Toolbar Button ..... 202
Photo Viewer Options ..... 202
Photograph View Window ..... 202
Location Selection ..... 203
Photograph Selection ..... 203
Photograph Attributes ..... 204
Copy Photograph to Clipboard ..... 204
Add Location ..... 204
Add Photo ..... 205
View Comments ..... 206

**1D Habitat Viewer Plug-in ..... 208**
Parts of the 1D Habitat Viewer Plug-in ..... 210
Graph Window ..... 210
Print Button ..... 210
Export Button ..... 210
Database Connection Button ..... 211
Habitat Station Selector ..... 212
Result Data Set Selector ..... 213
Cross Section Selector ..... 213
Other Features ..... 213
PHabSim 1D Data Visualization Tool Graphs ..... 214
Physical ..... 214
Stage Discharge ..... 215
Longitudinal Profile ..... 217
HSC ..... 218
Habitat ..... 219
Reach WUA ..... 220
Species Comparison ..... 221
Discharge Comparison ..... 222
Surface Plot ..... 222

**2D Habitat Viewer Plug-in ..... 224**
Parts of the 2D Habitat-Viewer Plug-In ..... 225
Site Selector ..... 225
Unit Selector ..... 225
Viewer Tabs ..... 226
Graph Window ..... 226
2D Data Viewer Graph Tabs ..... 226
Physical ..... 226
WUA ..... 230
HSC ..... 231
Fish Distribution ..... 232
Miscellaneous Data ..... 234

**Macroinvertebrate Data Viewer ..... 236**
Viewing Macroinvertebrate Data ..... 236
Macroinvertebrate Data Viewing Options ..... 237
Viewing Raw Data ..... 237
Size Distribution Plots ..... 238
Macroinvertebrate Density Plots ..... 239
Viewing Sample Information ..... 240

**Well Log Data Viewer Plug-in ..... 241**
Viewing well graphs ..... 241
Selecting Well Graph Points from the Map ..... 242
Viewing Well Graphs via the Well Log Data Viewer Toolbar Button ..... 242
Well Graph viewing options ..... 242
Viewing a Single Well Graph ..... 243
Viewing Multiple Well Graphs ..... 243
Exporting Graphs ..... 244
Viewing Well Report Documents ..... 245
Selecting Well Report Document Points from the Map ..... 245
Viewing Well Report Documents via the Well Log Data Viewer Toolbar Button 246
Well Report Document Viewing Options ..... 246
Viewing Entire Well Report Document Image ..... 246
Viewing Enlarged Image of Well Report Document ..... 247
Exporting Documents ..... 247

**Watershed Characterization Report Generator ..... 248**
Opening a MapWindow Project ..... 248
Changing the Database Configuration ..... 249
Starting the Report Generator plug-in ..... 251
Generating a Report ..... 251

**Appendix A: Importing Data into the Master Server ..... 256**
Editing Data Manually ..... 256
Importing Data Automatically ..... 259

**Appendix B: Importing 1D and 2D data from PHABWin_2002 ..... 263**
Appendix B-1: Importing 1D data from PHABWin_2002 ..... 263
Appendix B-2: Importing 2D Data into the 2D Habitat Viewer ..... 264
Appendix B-3: Using the 2D Data Importer ..... 272

**Appendix C: Extrapolation Program User Manual ..... 283**
Extrapolation Program ..... 283
Habitat Prediction Program ..... 292
Appendix C-1: Limitations of the Approach ..... 298

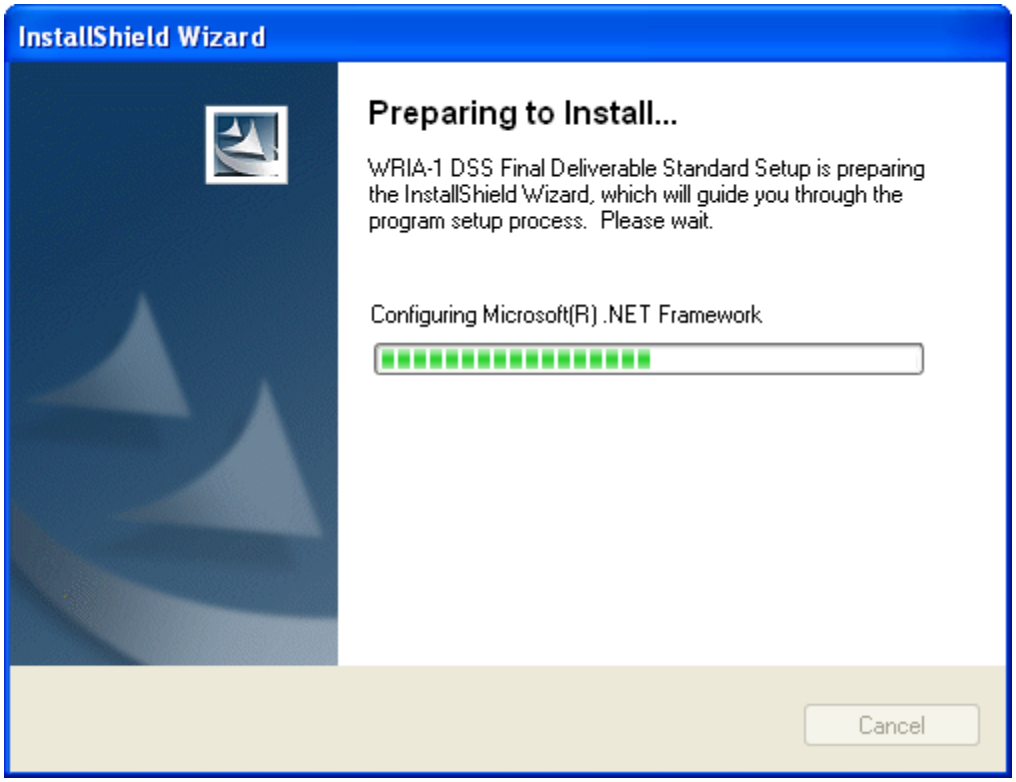
# Installing the WRIA-1 DSS

## Starting the Installation

To begin the installation, close all other programs.

Insert the first CD (labeled Disk 1). If the installation does not start within a few moments, double-click on the "My Computer" icon on your desktop. In the window that appears, double-click the CD-ROM drive. This should start the installation.

The following screen will appear once installation has successfully started:

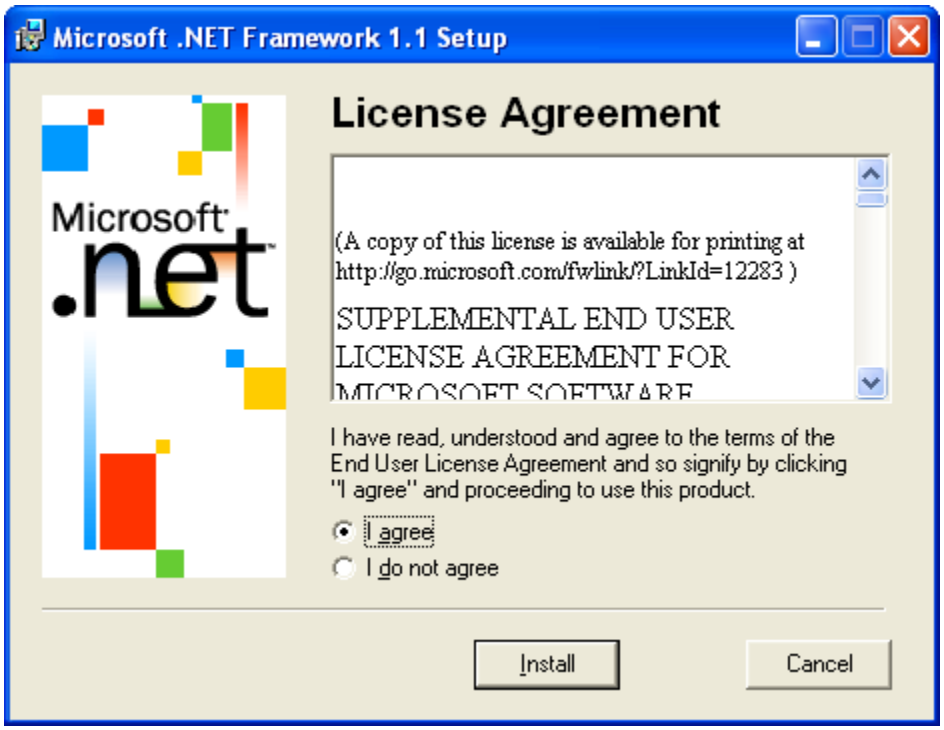

InstallShield Wizard
Preparing to Install...
WRIA-1 DSS Final Deliverable Standard Setup is preparing the InstallShield Wizard, which will guide you through the program setup process. Please wait.
Configuring Microsoft(R) .NET Framework
Cancel

## .NET Framework Installation

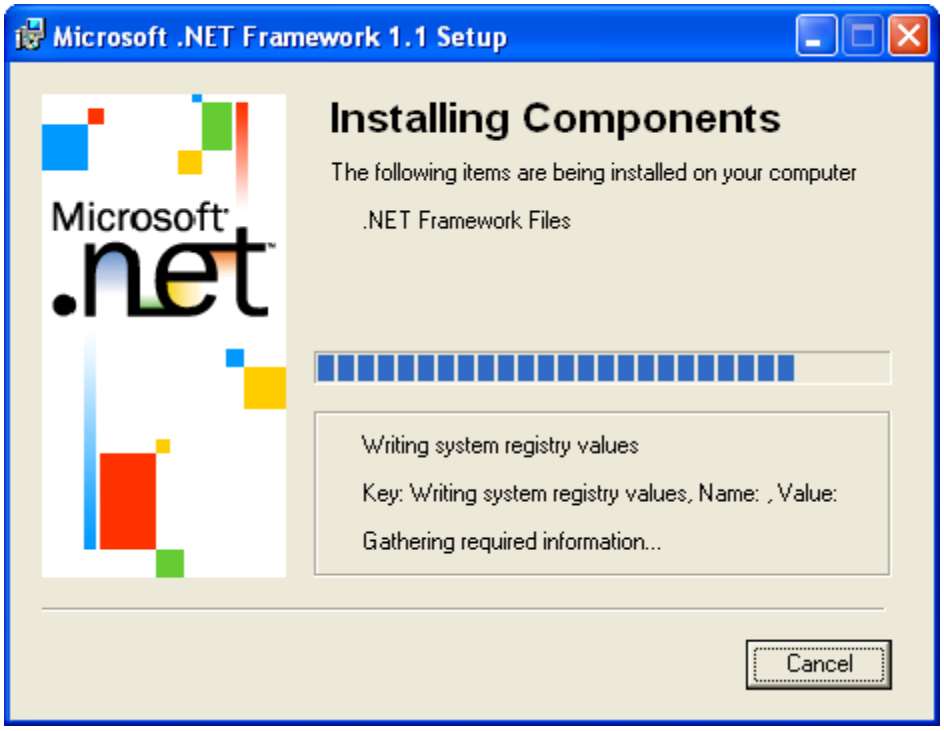
The next step is to install the .NET Application Framework. Much of the WRIA-1 DSS is based upon this component. This next step may not be necessary if your computer already has this component installed.

If the installation determines this step to be necessary, you will see the screen below. If you see a different screen, skip to the section labeled **Installation Welcome**.

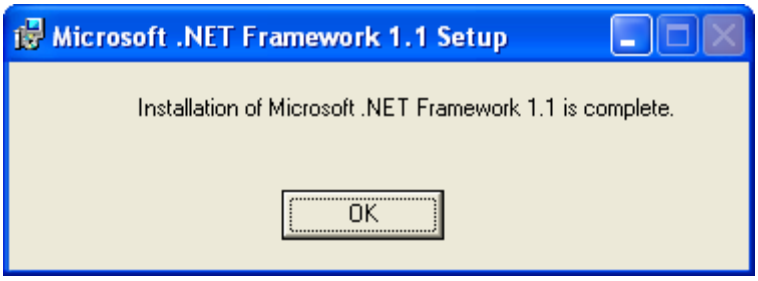
At this screen, browse through the License Agreement if you wish, then ensure that "I Agree" is selected. Press the **Install** button.


Microsoft .NET Framework 1.1 Setup
License Agreement
Microsoft
.net
(A copy of this license is available for printing at http://go.microsoft.com/fwlink/?LinkId=12283 )
SUPPLEMENTAL END USER LICENSE AGREEMENT FOR
I have read, understood and agree to the terms of the End User License Agreement and so signify by clicking "I agree" and proceeding to use this product.
I agree
I do not agree
Install
Cancel

After pressing the Install button, the .NET Framework will begin to install. You will see the following screen as this is happening:

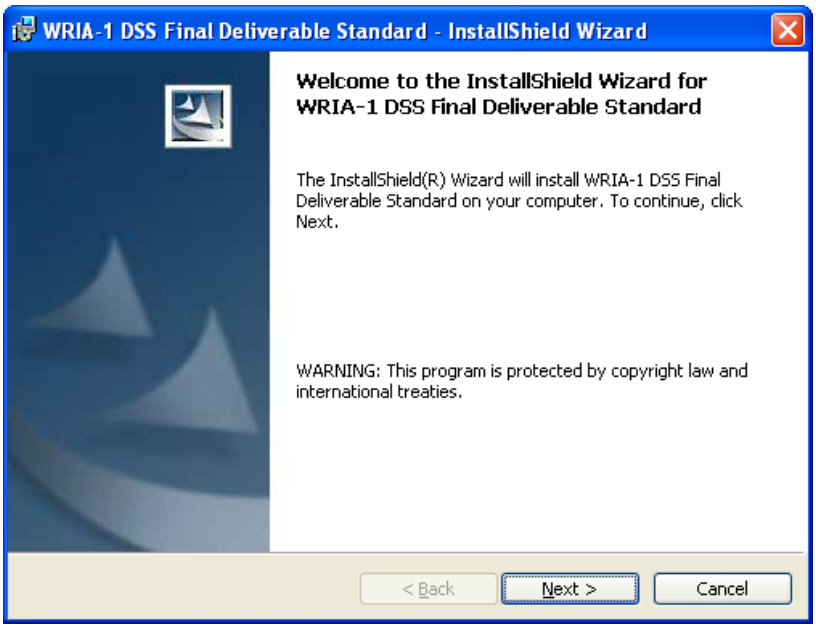

Microsoft .NET Framework 1.1 Setup
Installing Components
The following items are being installed on your computer
.NET Framework Files
Writing system registry values
Key: Writing system registry values, Name: , Value:
Gathering required information...
Cancel

When the installation of the .NET Framework has completed, you'll see the notification window below. Press **OK** on this message window.

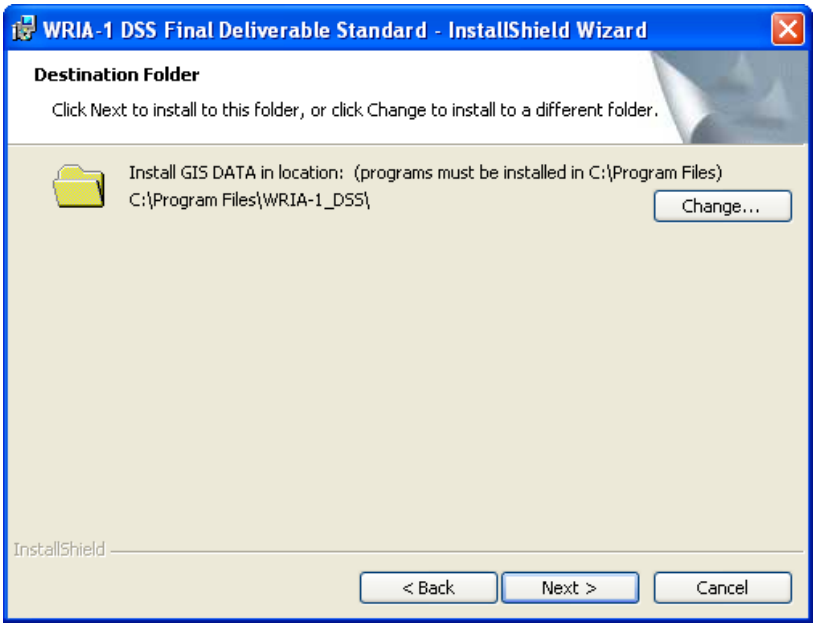


## Installation Welcome

Next, you'll be presented with a welcome screen. This simply tells you what program is about to installed, and gives you the opportunity to cancel. Select the **Next** button on this screen.


WRIA-1 DSS Final Deliverable Standard - InstallShield Wizard
Welcome to the InstallShield Wizard for WRIA-1 DSS Final Deliverable Standard
The InstallShield(R) Wizard will install WRIA-1 DSS Final Deliverable Standard on your computer. To continue, click Next.
WARNING: This program is protected by copyright law and international treaties.
< Back
Next >
Cancel

You will then be asked to choose a destination for the installation files. To use the default locations, press **Next**.


WRIA-1 DSS Final Deliverable Standard - InstallShield Wizard
Destination Folder
Click Next to install to this folder, or click Change to install to a different folder.
Install GIS DATA in location: (programs must be installed in C:\Program Files)
C:\Program Files\WRIA-1_DSS\
Change...
InstallShield
< Back
Next >
Cancel

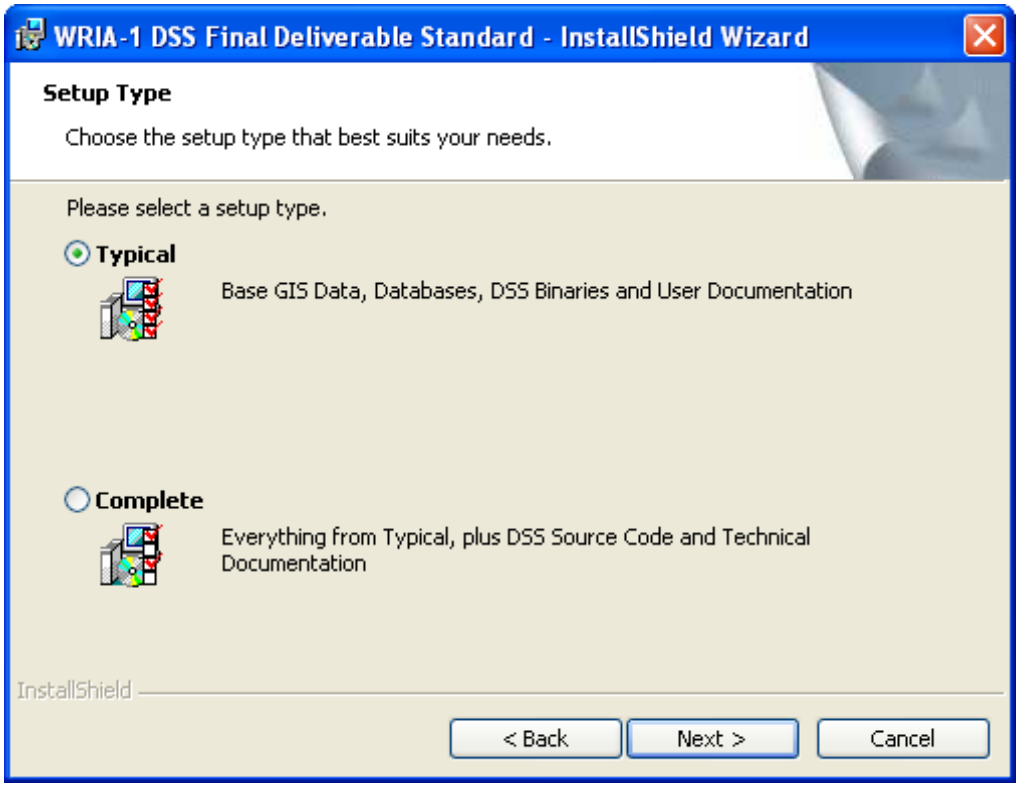
## Setup Type

If you are installing the Standard edition of the WRIA-1 DSS, you may skip this section. For the Standard edition of the DSS, clicking **Next** on the previous window will have started the installation process (see **Installing**).

If you're installing the Administrative edition of the WRIA-1 DSS, you will see the window below. This gives you the opportunity to select which components you would like to install. Choosing "Typical" will install the same software as the Standard edition, except that you will have access to the administrative tools in the DBMS (also known as LaunchPad).

Choosing "Complete" will install everything from "Typical" as well as the source code to the DSS components and the Technical Documentation.

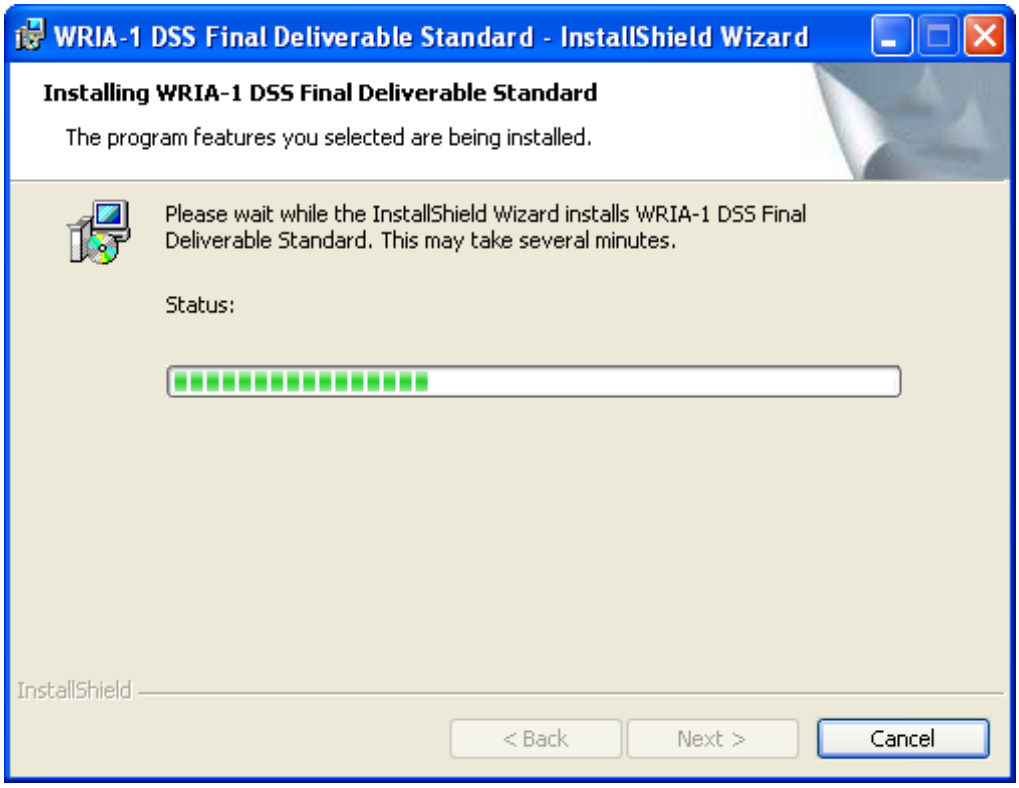
After selecting which setup type you would like, click the **Next** button.


WRIA-1 DSS Final Deliverable Standard - InstallShield Wizard
Setup Type
Choose the setup type that best suits your needs.
Please select a setup type.
Typical
Base GIS Data, Databases, DSS Binaries and User Documentation
Complete
Everything from Typical, plus DSS Source Code and Technical Documentation
InstallShield
< Back
Next >
Cancel

## Installing

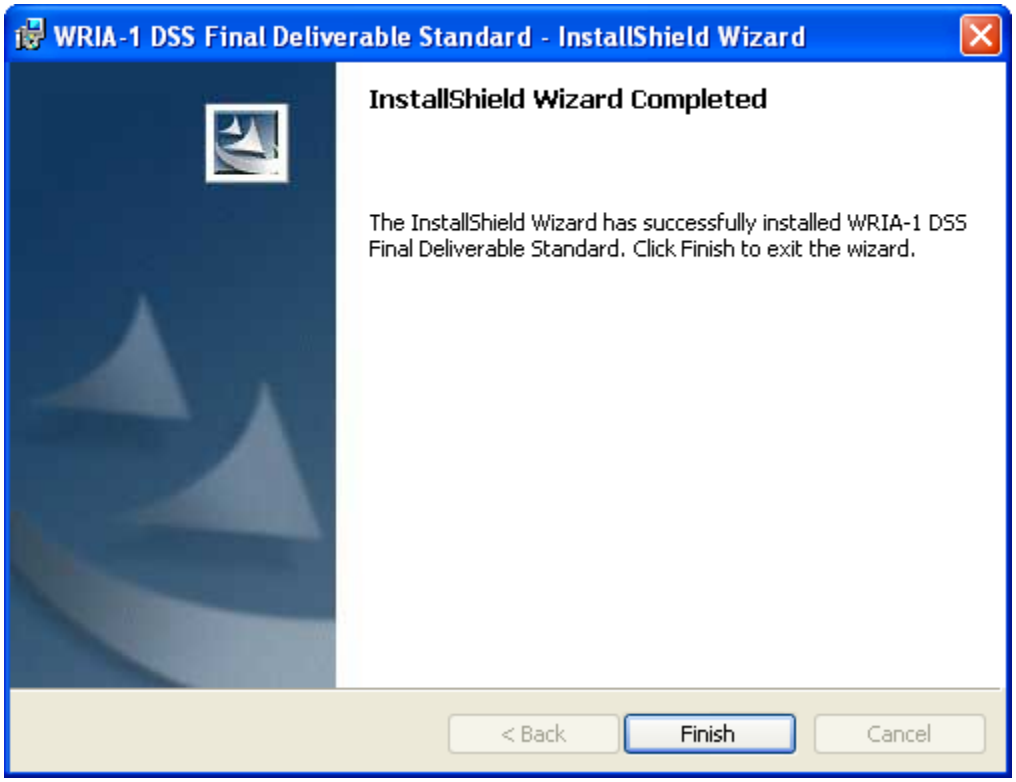
At this point, the installer will begin to actually install the software. This step may take some time, given the sheer volume of software and data being installed. You will see a meter in the center of the window; this does not indicate overall progress, but indicates the progress of the current step. The window you will see during the actual installation is shown below.

You will probably be prompted during this step to insert the next CD. When this appears, insert the next CD and click **OK** on the message window.


WRIA-1 DSS Final Deliverable Standard - InstallShield Wizard
Installing WRIA-1 DSS Final Deliverable Standard
The program features you selected are being installed.
Please wait while the InstallShield Wizard installs WRIA-1 DSS Final Deliverable Standard. This may take several minutes.
Status:
InstallShield
< Back
Next >
Cancel

## InstallShield Portion Complete

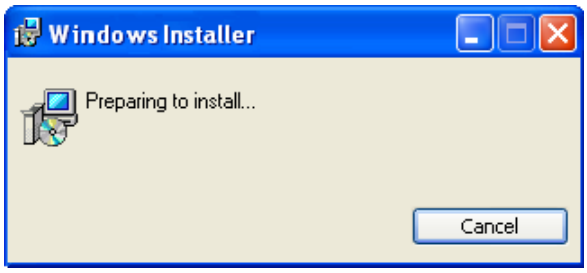
When the InstallShield portion of the installation has finished, you'll see the screen below. Click the **Finish** button to proceed to the next step.


WRIA-1 DSS Final Deliverable Standard - InstallShield Wizard
InstallShield Wizard Completed
The InstallShield Wizard has successfully installed WRIA-1 DSS Final Deliverable Standard. Click Finish to exit the wizard.
< Back
Finish
Cancel

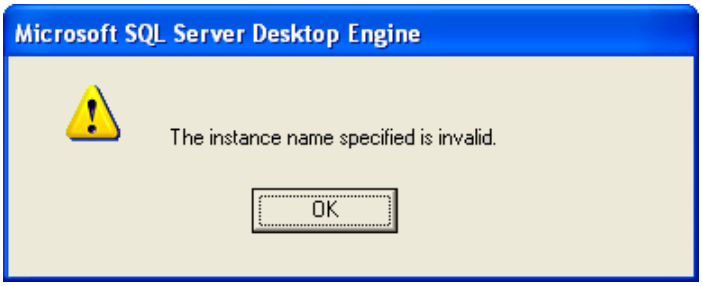
## Microsoft SQL Server Desktop Edition Installation

Microsoft SQL Server Desktop Edition is the database server utilized by the WRIA-1 DSS. This stores the main databases used by the Time Series Analyst and other plug-ins.

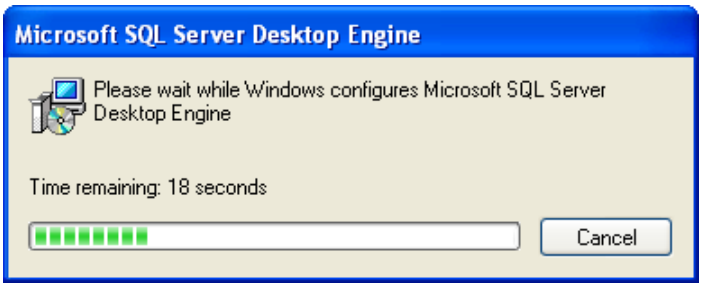
When you click **Finish** on the screen above, you'll see the window below, which informs you that the Microsoft SQL Server installation is beginning.



If the database server is already installed on your computer, you will receive the error message below. **This message may be ignored.** Simply click **OK** on the message box and proceed to the next step, Reboot.



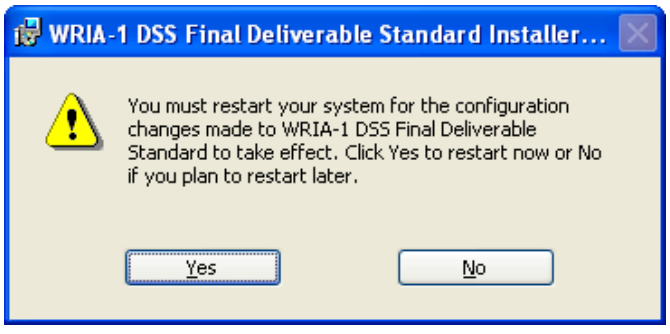

If the database server is not installed, then you'll see a progress meter with an estimated time remaining.


Microsoft SQL Server Desktop Engine
Please wait while Windows configures Microsoft SQL Server Desktop Engine
Time remaining: 18 seconds
Cancel

## Reboot

After the Microsoft SQL Server Desktop Engine has finished installing, or after you've clicked **OK** on the "Instance Name is Invalid" message, you'll see the screen below asking you to reboot your computer.

**You must restart your computer** to finish the installation of the WRIA-1 DSS. You can click **No** to restart later if you wish, but you must restart your computer before attempting to use the DSS.

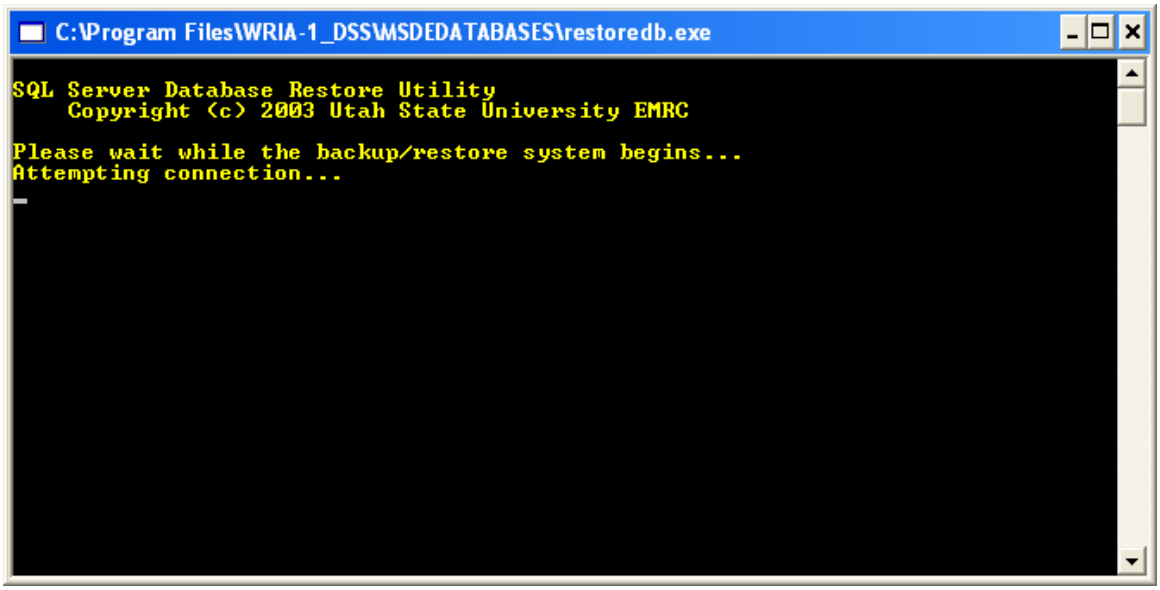

WRIA-1 DSS Final Deliverable Standard Installer...
You must restart your system for the configuration changes made to WRIA-1 DSS Final Deliverable Standard to take effect. Click Yes to restart now or No if you plan to restart later.
Yes
No

## Restoring Databases

After your computer reboots, you may need to log back in with your username and password. Please log in as the same user who began the installation (i.e., don't log in as "Bob" to start the installation, then after rebooting, log in as "Joe".)

When your computer starts back up, you will not immediately see your familiar Windows desktop. First, you'll see a small black window, as shown below. Inside this window, you'll see "Restoring Database FlowData…" followed by "Restoring Database ClimateData…" and so on, for four databases.

No action is required on your part here; simply wait for the process to complete. These databases are very large, so this step may take a few minutes.


C:\Program Files\WRIA-1_DSS\MSDEDATABASES\restoredb.exe
SQL Server Database Restore Utility
Copyright (c) 2003 Utah State University EMRC
Please wait while the backup/restore system begins...
Attempting connection...

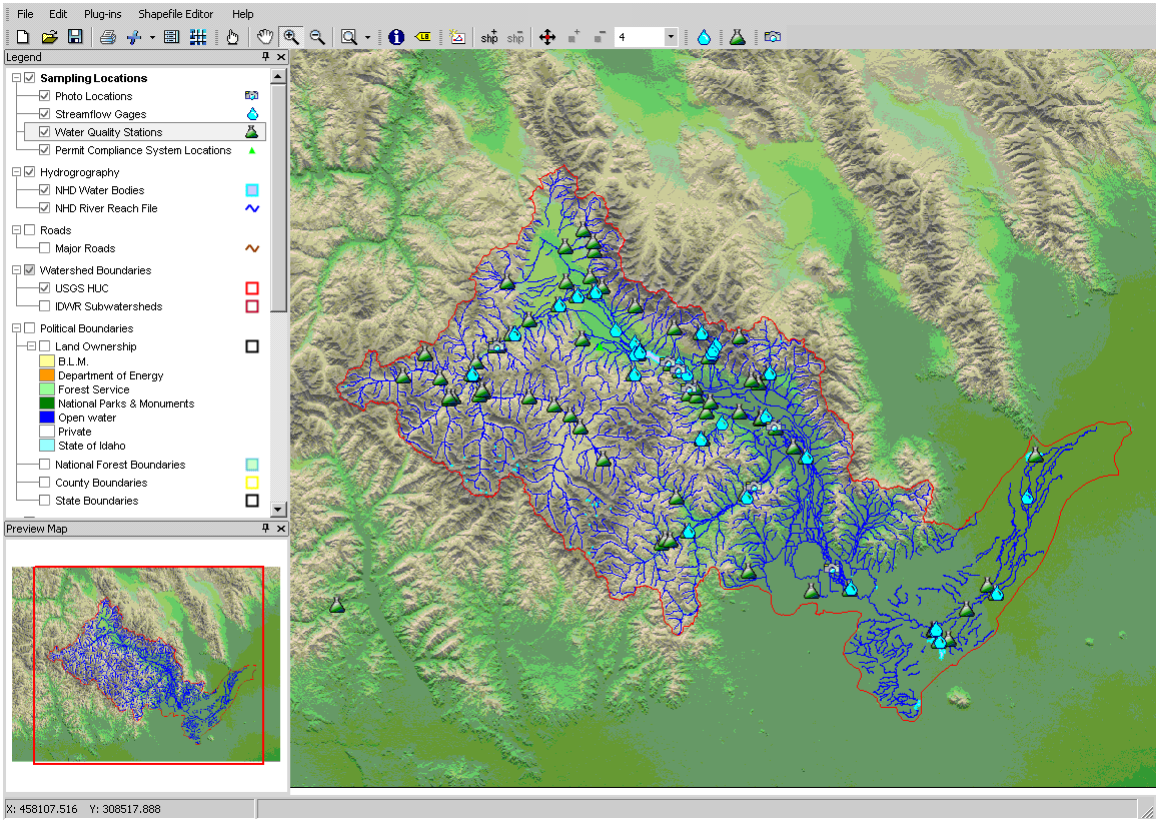
## Installation Complete

After the window above finishes, you'll be returned to your Windows desktop. Now the installation is complete – you may use the DSS.

The following sections contain detailed descriptions of the functions used within the DSS. It is not a systematic example of specifically what data to use, but rather an explanation of how the DSS works. Consequently, some windows may not display identical information for every possible option.

# MapWindow Watershed Information System

The MapWindow Watershed information system (WIS) is an extensible, GIS-based toolset that provides geospatial and temporal data visualization and analysis to aid watershed managers and decision makers. The WIS can be used for data visualization and analysis associated with Total Maximum Daily Load (TMDL) development/analysis, source water protection programs, and to aid watershed council members or other stakeholder groups in better understanding the data and information gathered by government agencies and other organizations.

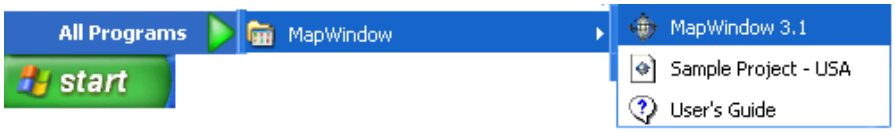



The base system includes an interactive map and legend, including typical GIS application functions, as well as a core set of plug-ins for analyzing water quality and quantity data, and digital photography. Because the base tool is inherently extensible, it allows for the implementation of custom plug-ins to address significant concerns in a specific area, and the inclusion of models and other data analyses that may be required. This allows users to create their own WIS to aid decision making for their area of interest.

# Overview of the MapWindow WIS Interface

## *Opening the MapWindow WIS*

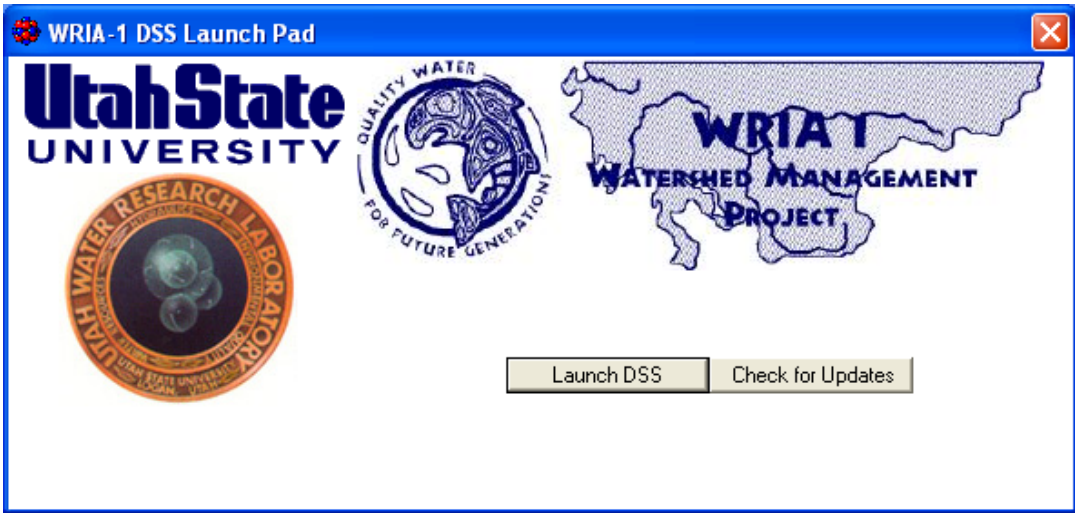
The installation of your WIS program should have placed a shortcut in the Windows **Start** Menu. To open your WIS program, click the **Start** menu (lower left corner of screen). From the **Programs** menu, select **MapWindow** and select the **MapWindow 3.1** program as shown below.




For more information about using this program, select the User's Guide from the list shown above.

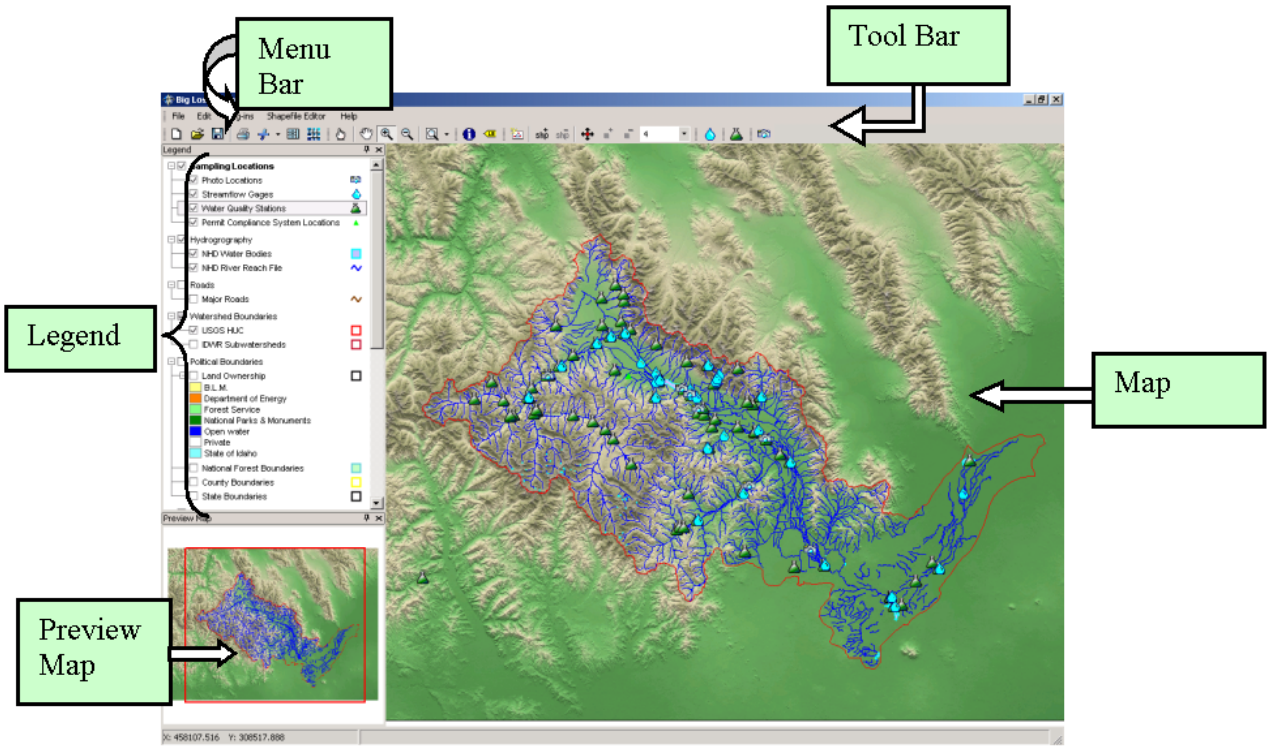
In the case of the Nooksack DSS, double-click the WRIA-1 DSS icon

on the desktop (the icon with a red and blue molecule). This will open the WRIA-1 DSS Launch Pad window as shown below. Press **Launch DSS** to begin.




The window contains three frames: (1) the Legend, (2) the Preview map, and (3) the Map. It also contains a Menu bar and a Tool bar. The following figure may help you understand the MapWindow WIS interface.

**Note:** Your WIS program may look different than the one below, depending on the configuration settings specified by the WIS developer.




## Parts of the MapWindow Interface

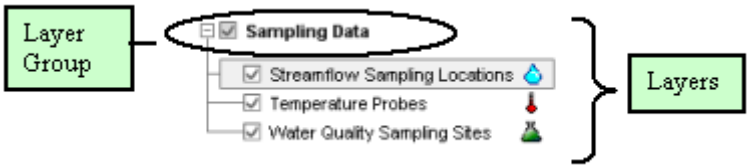
The following sections describe each of the major pieces of the MapWindow User Interface.

### *Map*

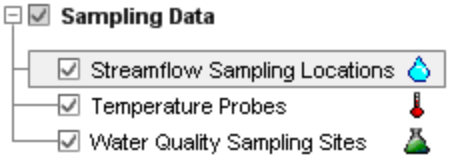
The map is the window in which roads, rivers, cities, county boundaries, water quality stations, habitat locations, and other spatial data are displayed.

### *Legend*

The legend displays *layer groups*, each of which consists of one or more *layers*. A layer is a coverage of spatially located points, lines or polygons that represent geographical information or site locations. Each layer may have attribute data associated with it; for example, a river reach layer may contain attribute information regarding the length, average flow, elevation, and reach geometry for each feature. Each layer in the legend can be displayed in the Map view.



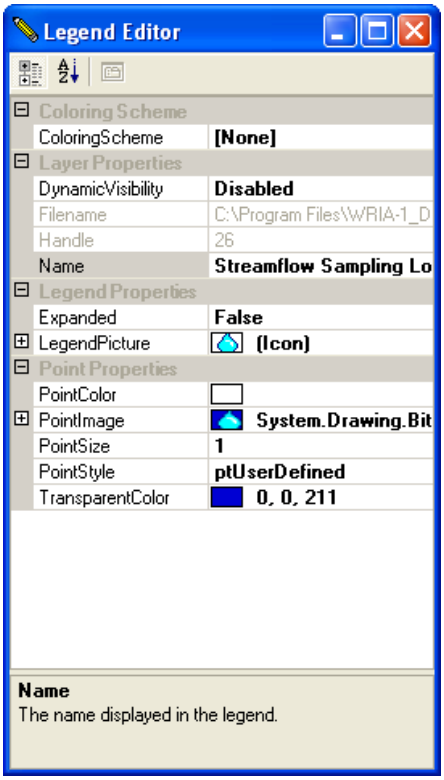

In the legend, you can turn 'on' and/or 'activate' layers. A layer is 'on' if it has a check mark in its corresponding box. A layer is 'active' if it has a box surrounding it and is gray. In the following example, the *Water QualitySampling Sites, Temperature Probes,* and *Streamflow Sampling Locations* layers are 'on,' but only the*Streamflow Sampling Locations* layer is 'active' (there can be only one 'active' layer at a time).



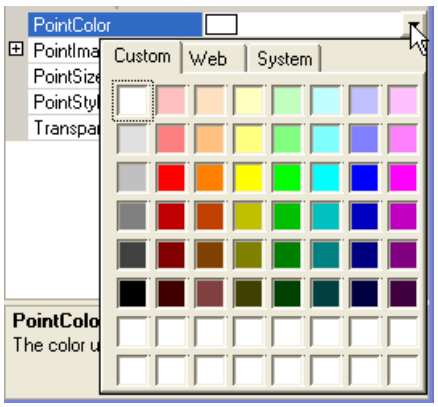

You can turn an entire group of layers 'on' by checking the box beside the layer group name. Similarly, you can turn an entire group of layers 'off' by un-checking the box beside the layer group name. If you want to display some layers in a layer group, but not others, you can turn 'on' and 'off' individual layers within each layer group. To check or un-check a layer, simply click in its corresponding box. When a layer is checked, it is displayed in the map.

### *Legend Editor*

The Legend Editor is a tool that allows you to change the way a layer is displayed. To open the layer editor, double click on the layer of interest in the legend. The following form will open.



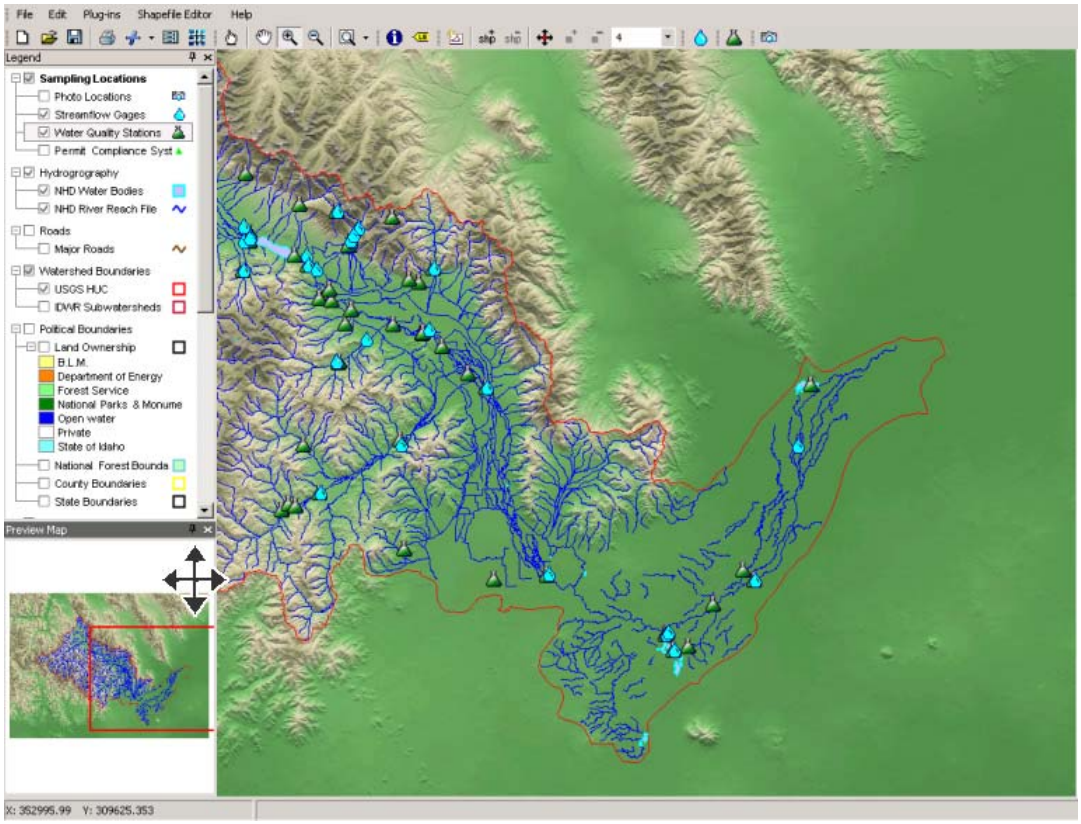

The Legend Editor contains five sections: Coloring Scheme, Display Properties, Layer Properties, Legend Properties, and Map Bitmap. Each section has a list of one or more properties. To change a property, click on its corresponding box at the right. A pull-down menu or dialogue box will become available that will allow you to make changes:




You can collapse a section by clicking its corresponding minus sign box, , at the extreme left of the form. You can expand a section by clicking on the corresponding plus sign box, .

### *Preview Map*

The Preview Map allows you to change the extents of the Map view. The red rectangle in the Preview Map shows the extents of the current Map view in relation to the overall project extents. To change the current extents via the Preview Map, click in the center of the red rectangle in the Preview Map and drag it to the desired position. Your Map view will automatically zoom to the updated extents:




You can change the way the Preview Map is displayed by turning on/off layers in the legend, zooming the Map view to the desired extents, and then right-clicking inside the Preview Map and selecting **Update Preview**. Upon being updated, the Preview Map will mimic whatever is being displayed in the Map view.

### *Select Feature Tool*

The select feature tool is used to select one or more features of the active layer. To use this tool, click on the Select Feature button, . To select one or more features, either point and click with your mouse (hold down the SHIFT key to select more than one feature), or draw a box surrounding the features. The feature(s) will turn yellow when selected:

***Zoom Tools***

The tool bar contains several zooming tools. The following tool buttons are available:

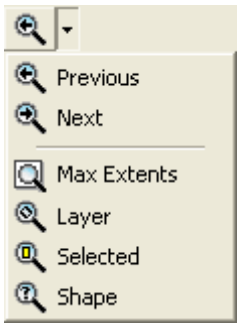
The Pan button allows you to drag your view in any direction without changing the scale of the map.

To zoom to a smaller area, click the Zoom In button, and then click on the map. The Map scale will be adjusted and the view will be centered at the point on which you clicked. In addition, you may click and drag a rectangle on the map and the map will be zoomed to the selected rectangle.

To zoom to a broader area, click the Zoom Out button, and then click on the map. The scale will be adjusted and the view will be centered at the point on which you clicked.

If you click on the magnifying glass portion of this button (circled in red), the map will zoom to the maximum extents of all visible layers.

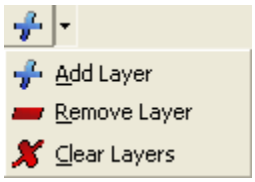
Clicking on the arrow portion of this button, located to the right of the magnifying glass, a list of options will become available:



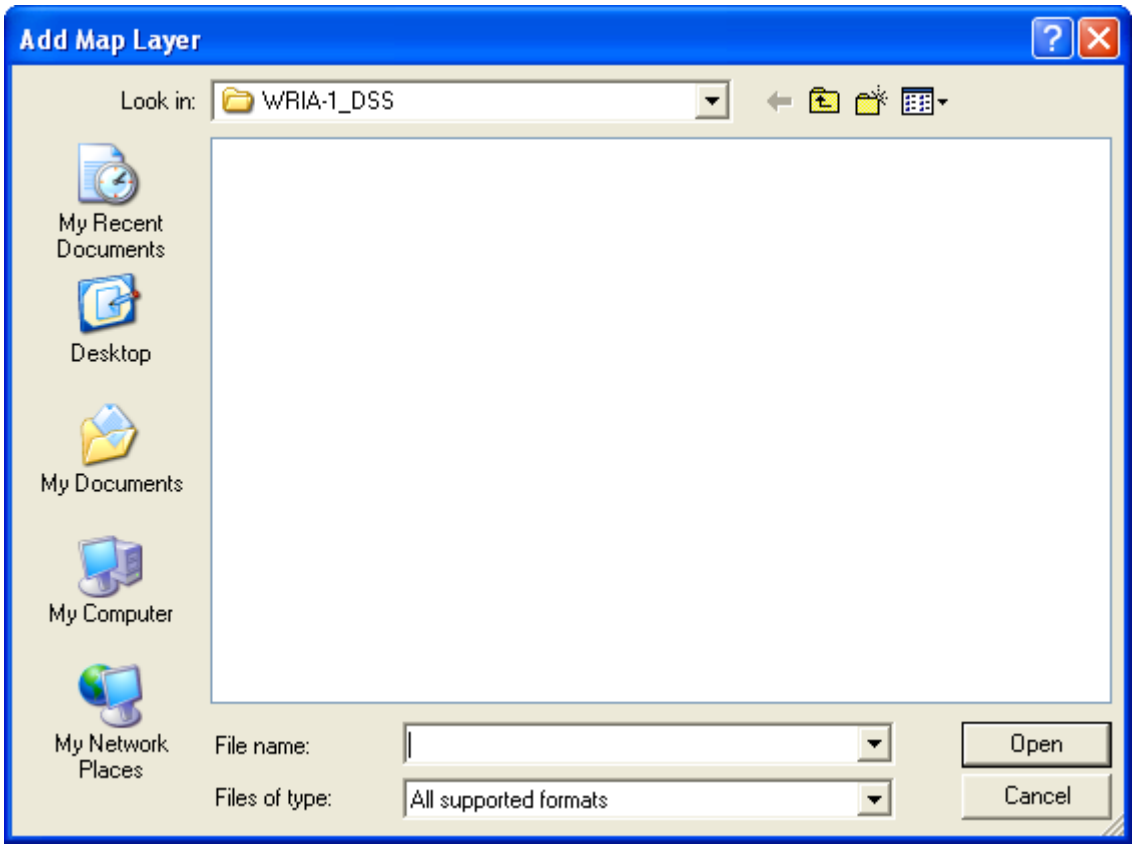

- Zoom to **Previous** changes the Map view extents to those of the previous zoom extents.
- Zoom to **Next** changes the Map view extents to those of the next view (only available if you have clicked Previous)
- Zoom to **Max Extents** changes the Map view extents so that you can view each layer in your project completely (it doesn't matter if the layers are 'on')
- Zoom to **Layer** zooms to the extents of the active layer
- Zoom to **Selected** zooms to the feature(s) you have selected in the active layer
- Zoom to **Shape** allows you to select a shape index from the active layer to which to zoom

***Layer Tools***

Within the MapWindow WIS, you can add layers to and remove layers from your project using the Layer Tools button, . To add or remove a layer, click on the menu arrow located on the right portion of the Layer Tools button. A list of options will become available:




If you select Add Layer, the Add Map Layer window will open:

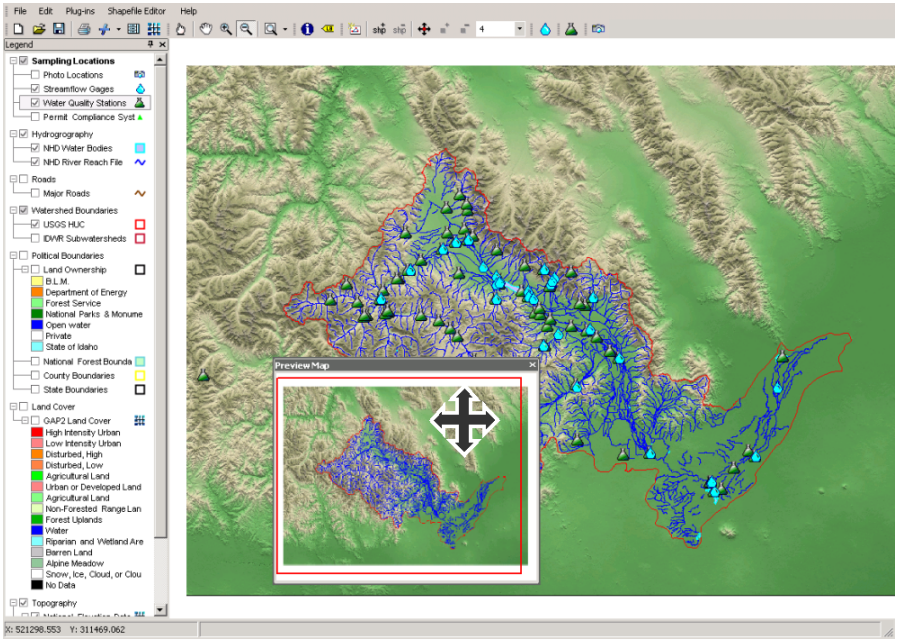



With this window, you can navigate to and select a new layer to add to your project.

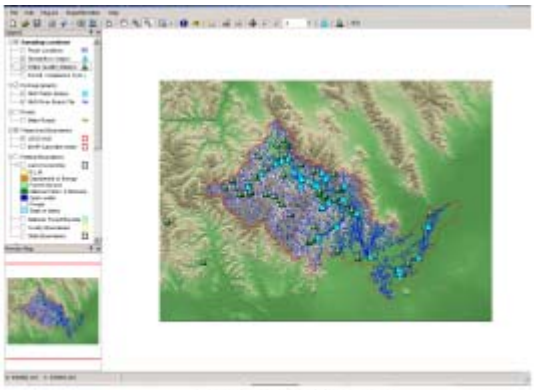
If you select **Remove Layer**, the layer that is *currently selected* will be removed from your project. If you select **Clear Layers**, ***all layers will be removed from your project!***

## Customizing the MapWindow WIS Interface

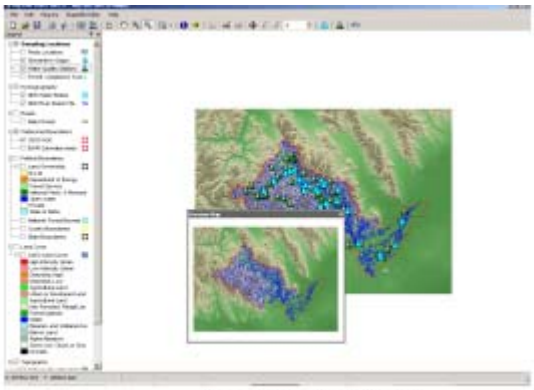
You can customize the MapWindow WIS interface to meet your preferences by moving and/or resizing the Legend and the Preview Map. To move a frame, click on the title bar (make sure the pointer changes to crossbars) and drag the frame to a new location:




You can change your view layout to either *docked* or *floating*. The view is docked when each frame is separate (no overlapping). The view is floating when one or more frames 'float' above another frame or frames:

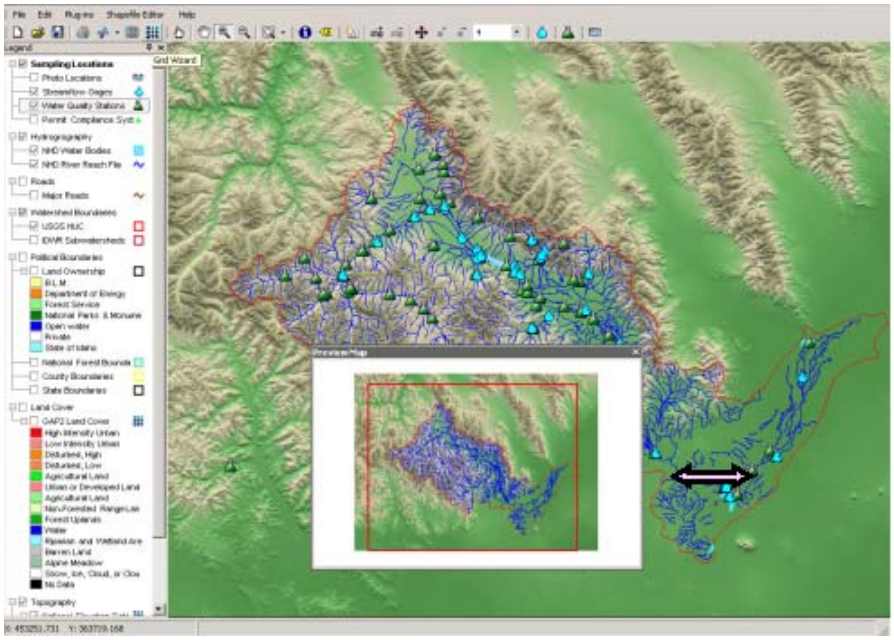


Docked Frames



Floating Frames

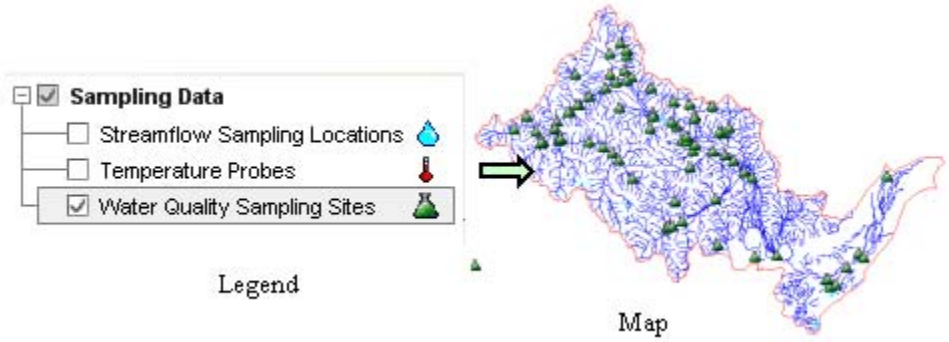
To resize a frame, move your mouse pointer to an edge of the frame and then click and drag the boundary until the frame is the desired size.



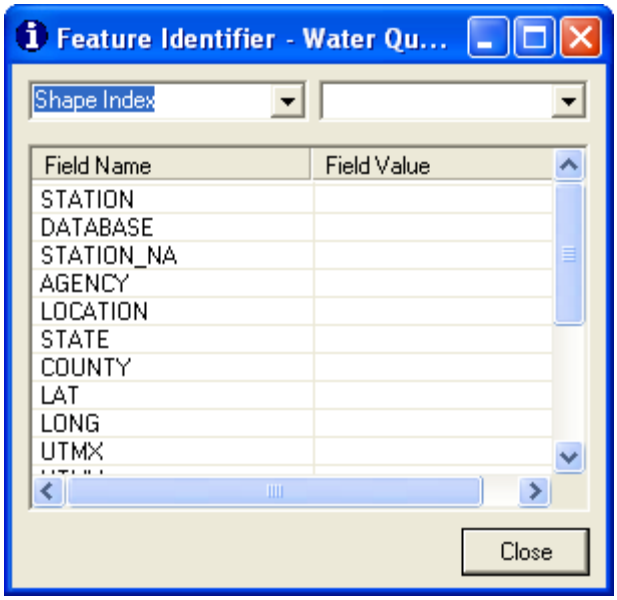
## Feature Identifier and Shape Labeler Plug-in

### *Feature Identifier*

The Feature Identifier tool allows you to click on a feature in the selected layer and view attribute information for the given feature. To use the Feature Identifier tool, first you must turn 'on' and activate the layer for which you would like information (recall that a theme is 'active' when it is gray and a box surrounds it and 'on' when it is checked):



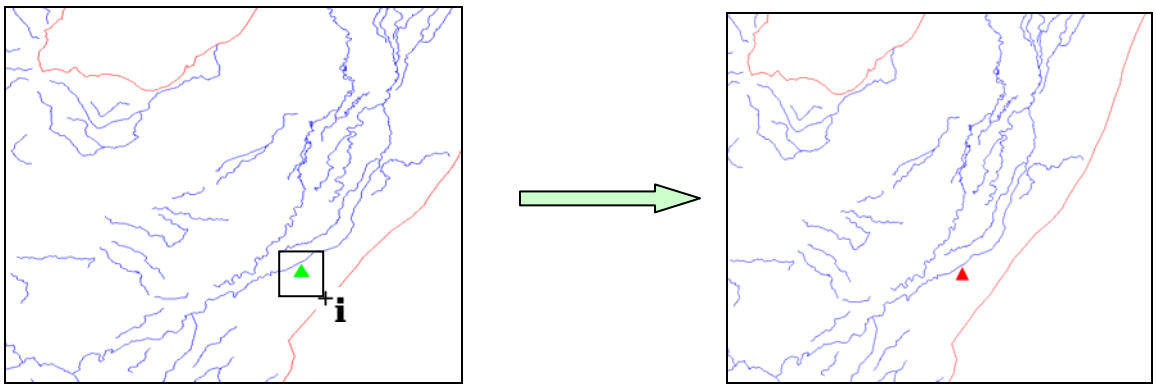

After you have activated a layer, click the Feature Identifier button, . The Feature Identifier window will open. This window contains a table with two columns: Field Name and Field Value. The items listed in "Field Name" correspond with the field names in the active layer's underlying attribute table. For the example above, the Feature Identifier table would look like the following:



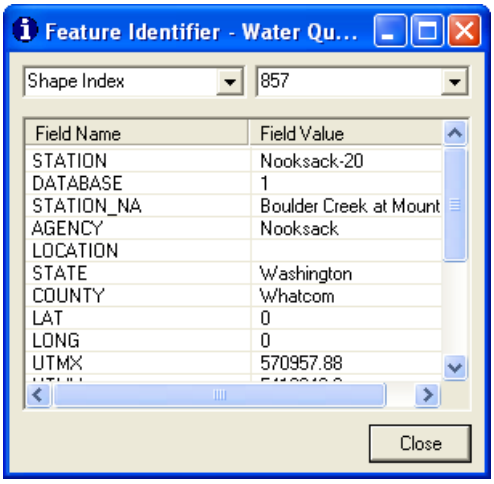

The values in the Field Value column will remain blank until you click on a feature within the selected layer.

**Note:** When the MapWindow WIS is in the Feature Identify mode, the cursor will appear as an "i" with cross bars in the upper left corner: +i

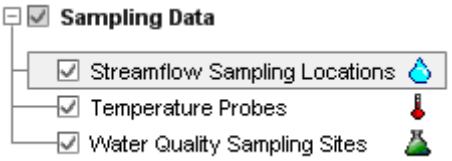
After you have clicked the Feature Identifier button, you can select a feature by either: (1) clicking on a point, line, polygon, etc. after positioning the crossbars of the cursor directly over the feature, or (2) clicking and drawing a box surrounding the feature or surrounding part of the feature(s). The selected feature will change color.

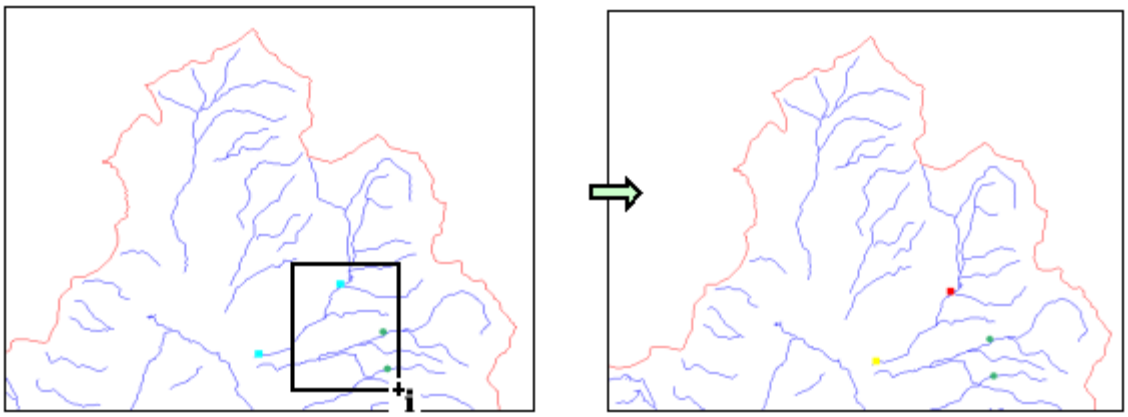



The Feature Identifier table will become populated with data that corresponds to the feature you selected:



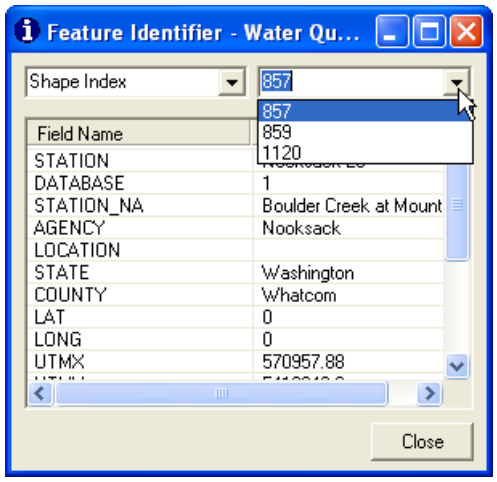

In the example above, we clicked on a feature in the map and then viewed its attribute information in the Feature Identifier window. This tool can also locate features on a map based on user-defined specifications. The user can activate a layer, click the Feature Identifier tool, and then select several features of the layer:



When more than one feature is selected with the Feature Identifier tool, only one feature's attributes will appear in the Feature Identifier table. The feature that appears in the table will change color in the Map View. The other selected features will remain unchanged.

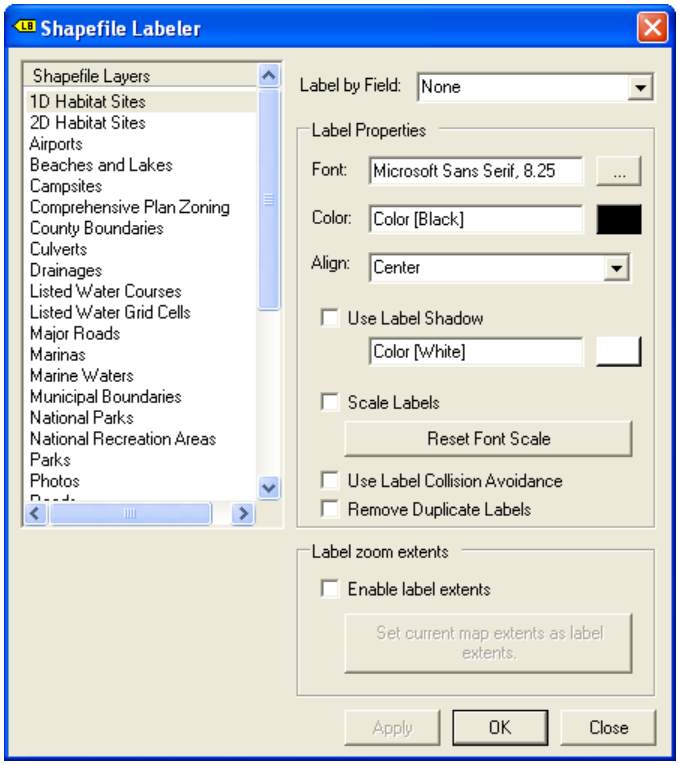
The Feature Identifier table includes two fields at the top of the window in which you can perform queries on the selected features. In the right box, you can select an attribute. In the left, you can select a corresponding attribute value:




In the example above, the Water Quality Sampling Site gage(s) that has a "Shape Index" of "857" will turn red in the Map view.

***Shape Labeler***

***The Shape Labeler allows you to add customized labels to the Map view. To open the tool, click the Shape Labeler button, . The following window will open:***



***In the Shapefile Layers list, select the layer for which you would like to add labels. The "Label by Field" menu will become populated with the layer's corresponding attributes:***

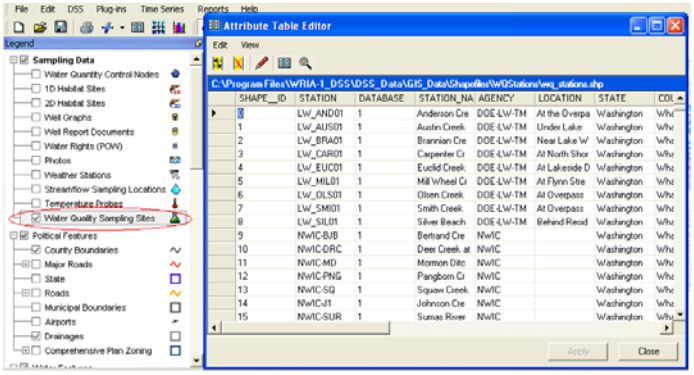
After you select an attribute by which the layer will be layered (for example, station name, ID number, or constituent type), you can customize the label's appearance by changing the text font, color, and alignment using the Label Properties fields:
To apply your labels to the Map view, click **Apply**.

The labels will appear in the Map view whenever the corresponding layer is displayed. To remove a layer's labels, click the Shape Labeler button, select the corresponding layer in the "Shapefile Labeler" list, and select **None** in the "Label by Field" list.

The Label Zoom Extents option allows you to set a map zoom level at which the labels will turn on. This is useful when labels on the map become too cluttered to read. You can set the label extents such that the labels do not turn on until you zoom in to a certain level on the map. To do this, click the Enable Label Extents check box. This will activate the label zoom extents option. Next, using the Zoom Tools, zoom the Map View to the desired zoom level at which you wish your labels to appear and then click on the Set Current Map Extents as Label Extents button. Press **Apply** once the desired zoom level has been set. This option can be disabled by simply un-checking on the Enable Label Extents check box.

## Attribute Table Editor Plug-in

The Attribute Table Editor Plug-in allows you to view and edit the attribute data associated with the data layers in the Map view. To view the attribute table associated with one of the layers, first make the desired layer active and then click on the Attribute Table Editor Plug-in icon on the toolbar. This will open the attribute table for the active theme.




The columns in the table can be resized to show their contents by hovering over the line between two columns until the resize cursor appears and then clicking and dragging the

column to its desired width. Clicking the field name at the top of the table sorts the field in ascending or descending order.

Information in each of the fields can be changed by clicking in the fields and editing the contents. Fields can be either added or deleted using the Add and Delete field buttons at the top of the form. Fields can be renamed using the Rename Field button . Records in the table can be selected by clicking on the grey bar at the extreme left of the table. In addition, once a record or group of records is selected in the table, you can use the Show Selected Records button to hide the records that are not selected. The last button on the Attribute Table Editor toolbar is the Zoom to Selected Shapes button. When clicked, this button zooms the Map view to the shape corresponding to the selected record in the attribute table that you were viewing.

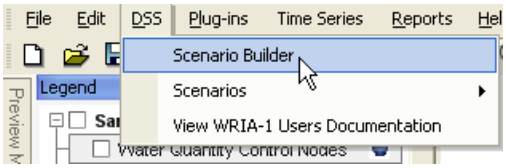
For additional help, see the User's Guide, accessible through the All Programs menu under the Start Menu as shown below.

# Scenario Builder
# (also commonly called Model Manager or DSS)

## Starting the DSS

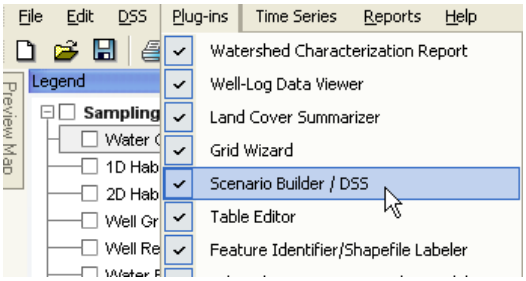
Double-click the WRIA-1 DSS icon on the desktop (the icon with a red and blue molecule). This will open the WRIA-1 DSS Launch Pad window as shown below. Press **Launch DSS** to begin.

This will bring up a MapWindow and the Nooksack project. One of the menus in MapWindow will be labeled **DSS**, as shown below:




Select the **Scenario Builder** option to start the Model Manager. Note that this tool goes by three different names, which are used interchangeably throughout all literature and software: Model Manager, Scenario Builder, and DSS. This last term is best avoided, since the term is also used to refer to the complete package of software and data.

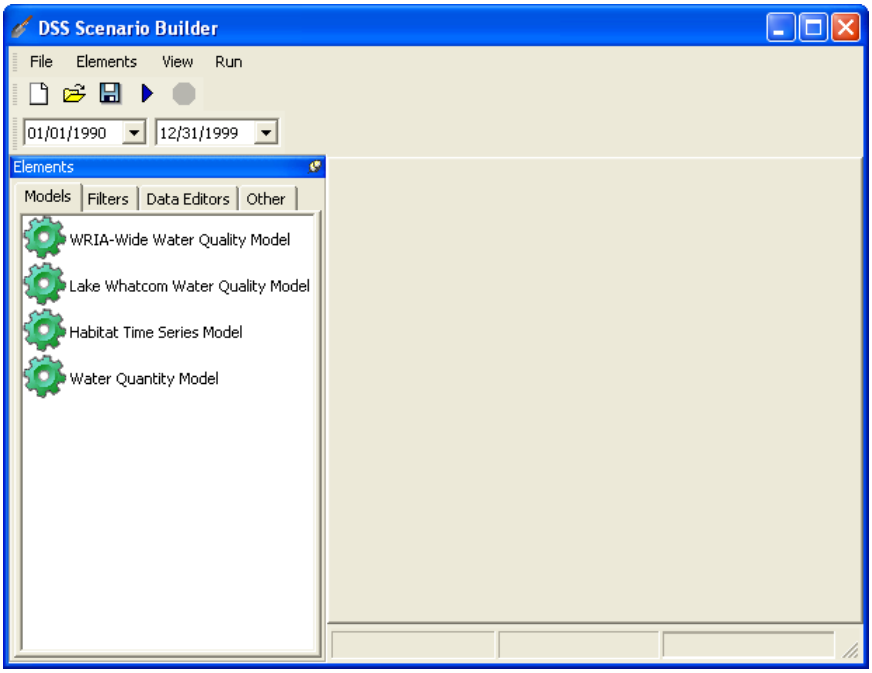
If this menu does not appear, select the **Plug-ins** menu instead. You will see an option labeled **Scenario Builder / DSS**. If there is no check mark next to this item, click the item as shown below.



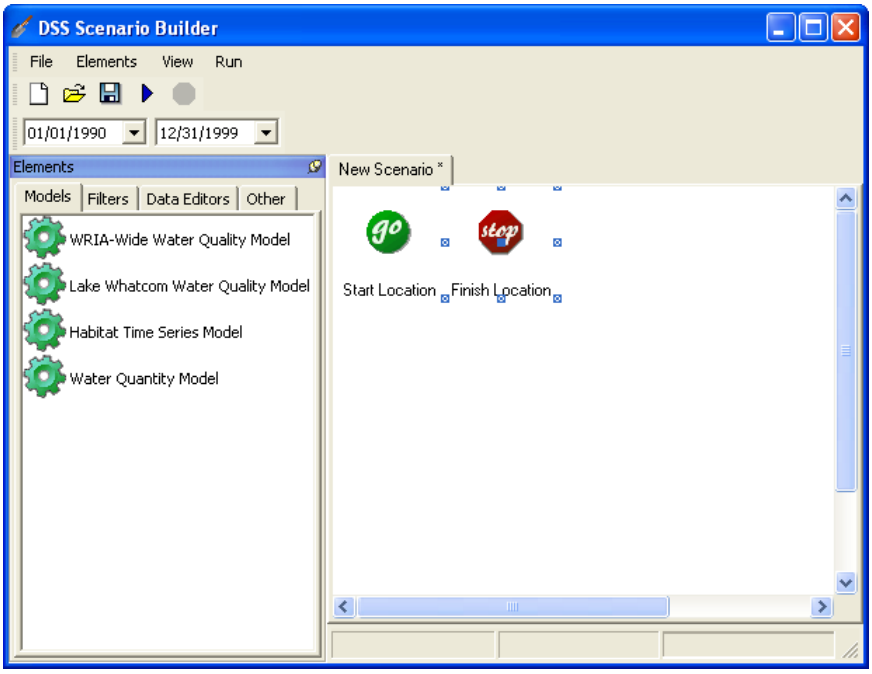

If the **Scenario Builder / DSS** option does not appear in the Plug-ins list, you do not have the Model Manager installed.

## Creating a New Scenario

When the scenario builder is first started, you will be presented with an empty screen as shown below.




In order to create a new scenario, click the white paper icon or select the **File** menu, then the **New** option. After creating a new project, you will see the screen below.



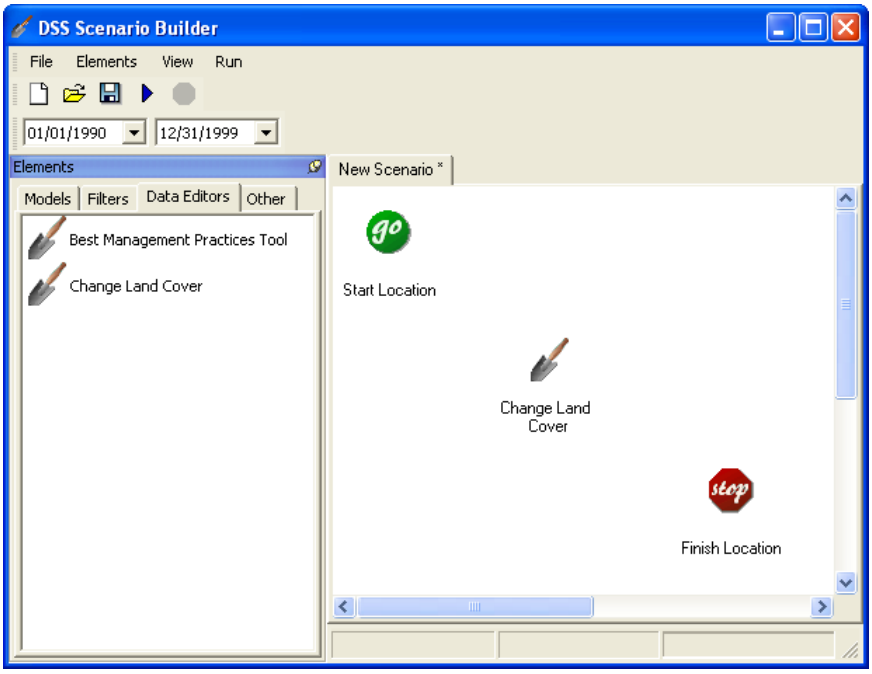

### *Adding Elements*

Next, you may select any of the tabs to the left (the "Element Toolbox"). Here, you will see model elements (also called scenario elements) listed. You may add one of these to your project either by double-clicking on the scenario element, or by dragging it from the toolbox into the scenario (the window with the Go and Stop icons).

You may add the same element to the scenario more than once, if your particular scenario requires this. As you add elements to the scenario, it is a good idea to rearrange them so that they fall in a logical place on the screen. You may move a model element by simply clicking on it and holding down the mouse key to drag it.

The screen below shows what the Scenario Builder looks like after adding the Land Cover Data Editor.

Note that this example shows the mechanics of setting up a scenario without including all the various specific configuration parameters necessary to run a real scenario.



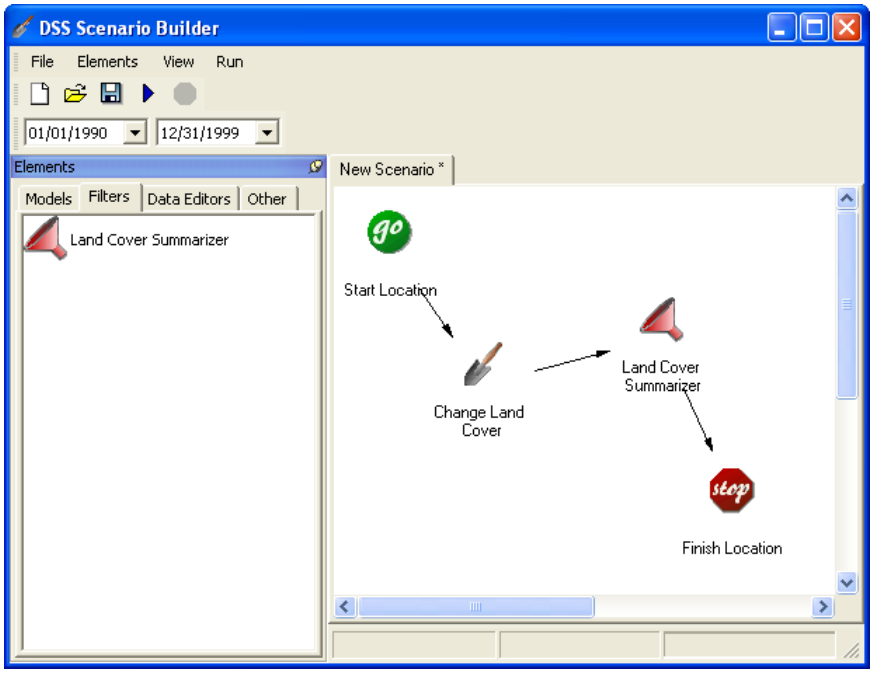

### *Connecting Elements*

Once all of the model elements you wish to use in your scenario have been added, they will need to be connected. This is done by clicking once on the first element to be connected – a box will appear in the very center of the element. Click and hold the mouse button down on this box and drag the mouse (while holding the mouse button down), and you will see a line come from the model element to your cursor – continue dragging the mouse pointer to the destination element and release the mouse. The elements will be connected by one directional arrow.

All model elements must be connected to form a path from the Start Location to the Finish Location. When the model is executed, the Scenario Builder will start at the "Go" icon and follow the arrow from it to the first element, which will then be executed. The Scenario Builder will follow the arrow from that element to the next element, process it, and so on until the Scenario Builder reaches the "Stop" icon.

Since elements are executed in sequence, the order in which elements are connected in a scenario matters. In this example, the changes to land cover will be applied before the land cover is summarized. Therefore, the land cover data editor must run before the land cover summarizer. There could be additional elements between the two, but since the edit takes place before the summary, that's all that really matters in the positioning of elements. As a rule of thumb, elements that produce input for another element must run before that element.

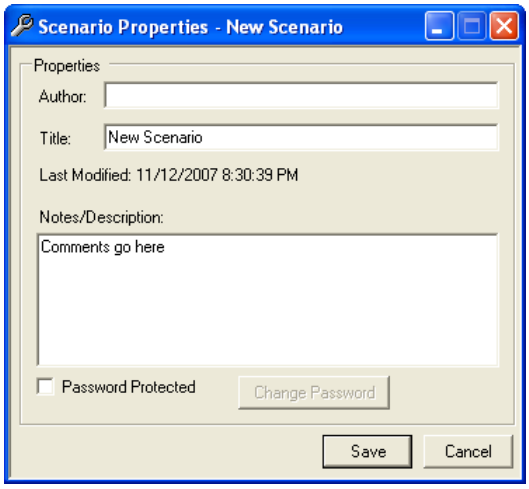
The screen below shows a scenario with the Land Cover Data Editor and the Land Cover Summarizer properly connected:




### *Viewing Properties and the Scenario Tab*

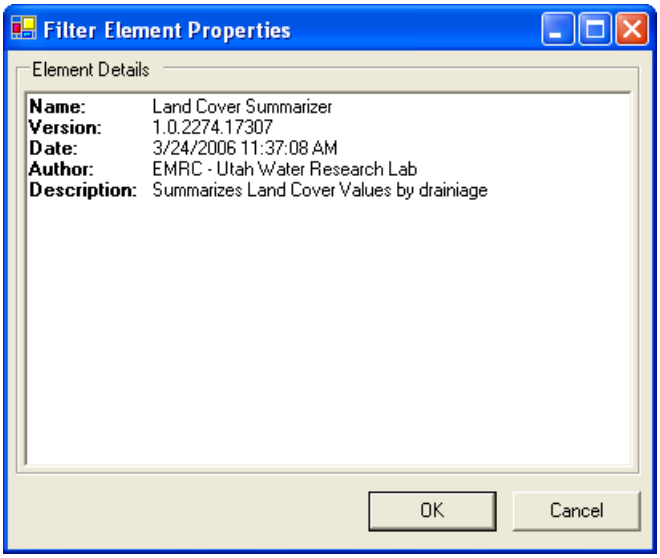
Each scenario has properties associated with it such as the scenario author, title and notes or descriptions. The scenario properties may be viewed and edited by either double-clicking on the tab above the scenario, or by selecting the **View** menu, then **Scenario Properties**.

A Scenario Properties screen is shown below. Any of the fields may be edited here, and the scenario can be password-protected. Click Save to continue, or Cancel to abandon any changes.



The tab that displays the name of the scenario will often have an asterisk (*) after the scenario name; this signifies that the scenario has not yet been saved or has been modified since it was last saved. Several scenarios may be open at once; each open scenario will have its own tab. Switching between scenarios is done by clicking the tab with the desired scenario's name.

Each element inside a scenario has properties associated with it, but these properties cannot be edited. The properties for an element can be viewed by selecting the element in the scenario (click on it once). Next, click on the **View** menu and select **Element Properties**. An example Element Properties screen is shown below.




### *Configuring Elements*

Many of the scenario elements must be configured before the scenario can be executed. Every scenario element will differ in what needs to be configured and how this is accomplished. A scenario element is configured by double-clicking on the element's icon in the scenario window (the window with the "Go" and "Stop" icons).

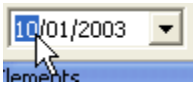
Double-clicking on a scenario element brings up the configuration screen for that particular element. Any configuration needed to run the selected element can be done using this configuration screen. For more specific configuration information, refer to the section in this document describing the particular element of interest. Once data and configuration parameters have been entered for an element, the information will be saved as a part of the scenario.

The various parameters available for each scenario element allows for multiple instances of that scenario element in the same scenario: each element is configured independently, so, for instance, a scenario could have several land cover summarizer elements, each configured to summarize from a different land cover grid.

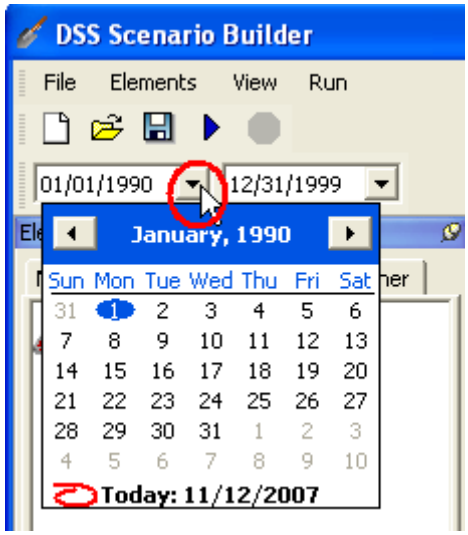
### *Setting the Date Range*

Some model elements will require a date range for processing, for example, a model analyzing stream flow data may need to know what date range of samples should be examined. Not all scenario elements use this; those that don't use this information will simply ignore it.

Setting the scenario date range is done by clicking on the date values and typing in new values as shown below.

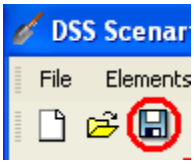



Also, the date ranges can be modified by clicking on the down arrow to the right of the text fields with dates in them as shown in the image below (circled in red). The leftmost date is the starting date, and the rightmost date is the ending date. Clicking on the arrow will cause the calendar to appear, which can be used to select a new date.



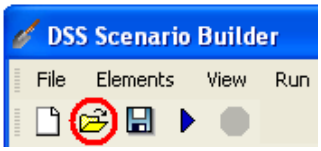

### *Saving the Scenario*

Scenarios are not saved to a file on disk, but rather are saved to the Scenario Builder's internal database. A scenario can be saved by clicking on the floppy disk icon, circled in red below:



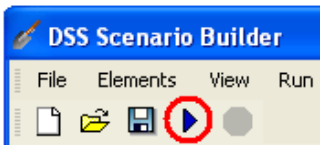

The scenario is also saved automatically whenever it is executed.

A scenario can be opened again by choosing the **File** menu, then **Open**, or alternatively by clicking the yellow folder icon located to the left of the save button, circled in red below.



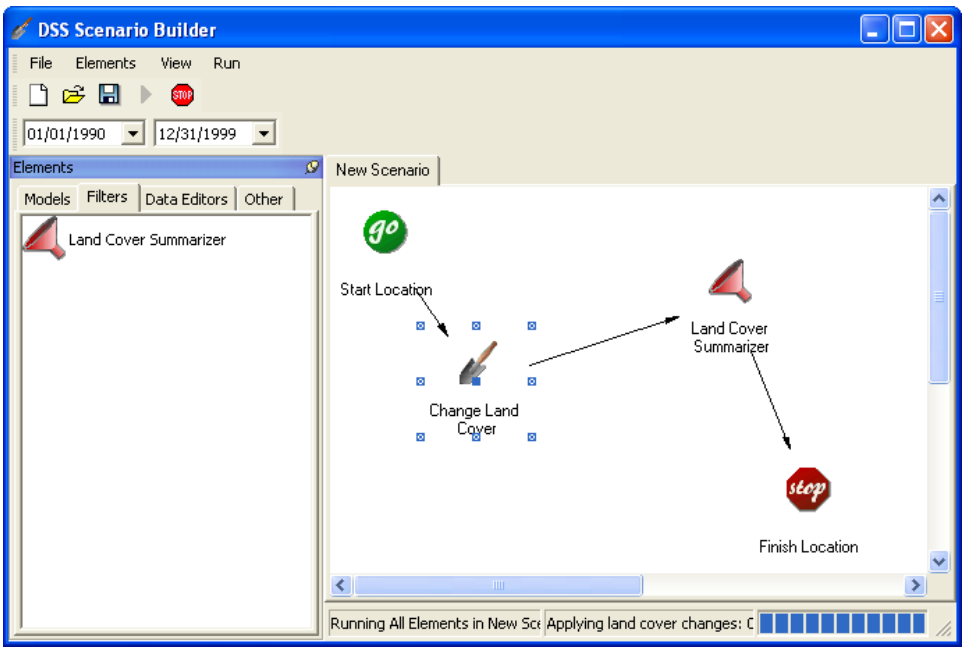

## Running the Scenario

A scenario may be executed in one of two ways; selecting the **Run** menu, then select **Run All Elements**, or by clicking the "Play" icon, circled in red below:




Notice that there is also a **Selected** option in the **Run** menu; this will run the currently selected scenario element only, ignoring all other elements in the scenario. This option should be used with caution, as scenario elements often rely on data produced by other elements in the scenario.

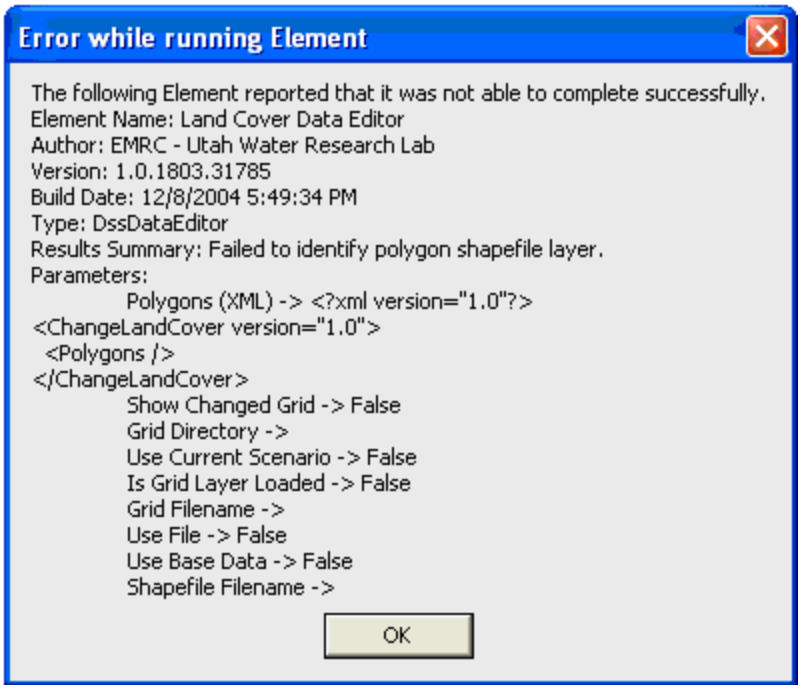
Once a scenario has started running, the screen will appear as shown below. Notice the outline around the element that is currently running, and the progress message and progress bar at the bottom of the screen.



If there are any errors in the configuration of the scenario, it will not run properly and an error message will appear.

### *Errors*

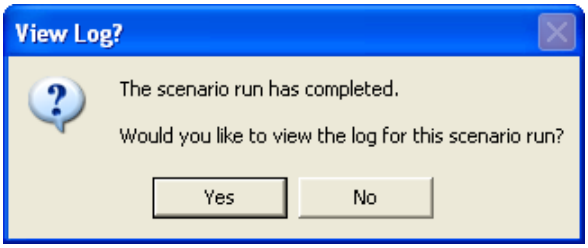
When running a scenario, one or more of the scenario elements may report errors during execution. This is often due to the user forgetting to configure an element, or improperly configuring an element (i.e., asking an element to use a nonexistent database). The error message which appears, though not visually appealing, is quite informative:




Reading this error message, it is visible that the error was caused by the Land Cover Data Editor. Scanning through the error message, notice the Results Summary, which says "Failed to identify polygon shapefile layer". This suggests the need to check the configuration of the model element (by double-clicking the element in the scenario builder) to ensure that all of the selected polygon shapefiles actually exist and are the proper files.

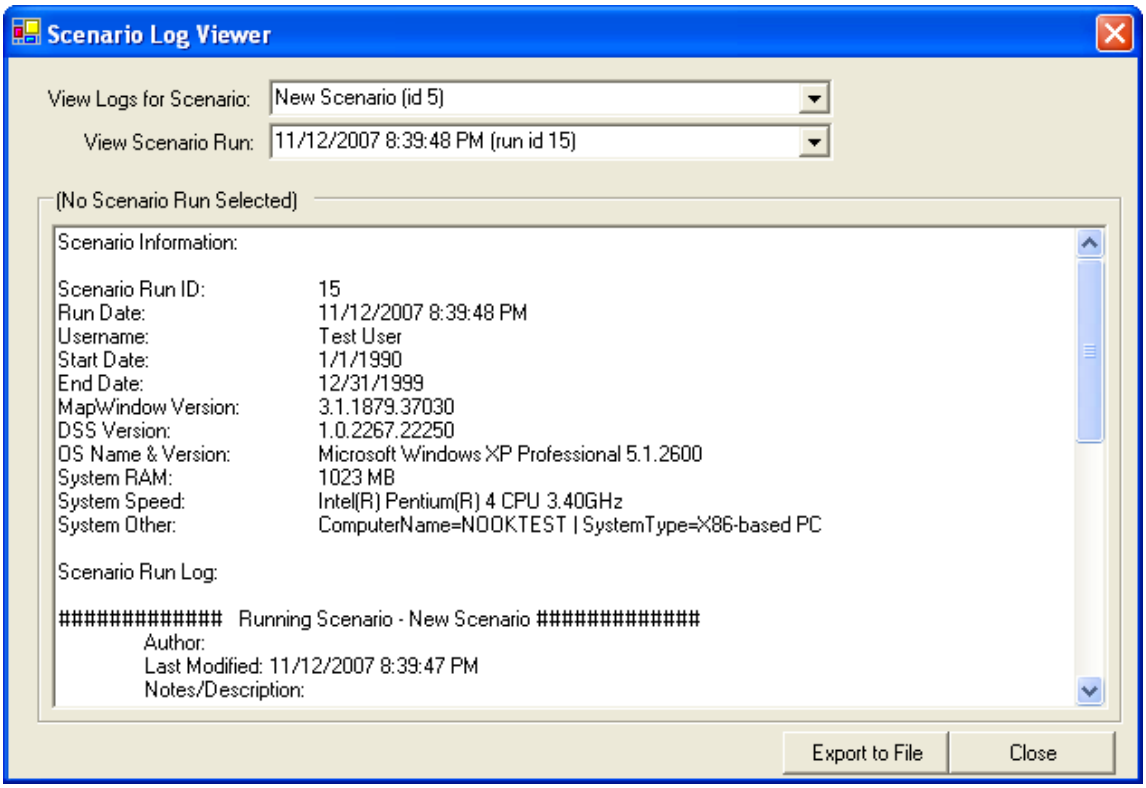
### *Scenario Run Log*

When a scenario has finished running the following window will appear.



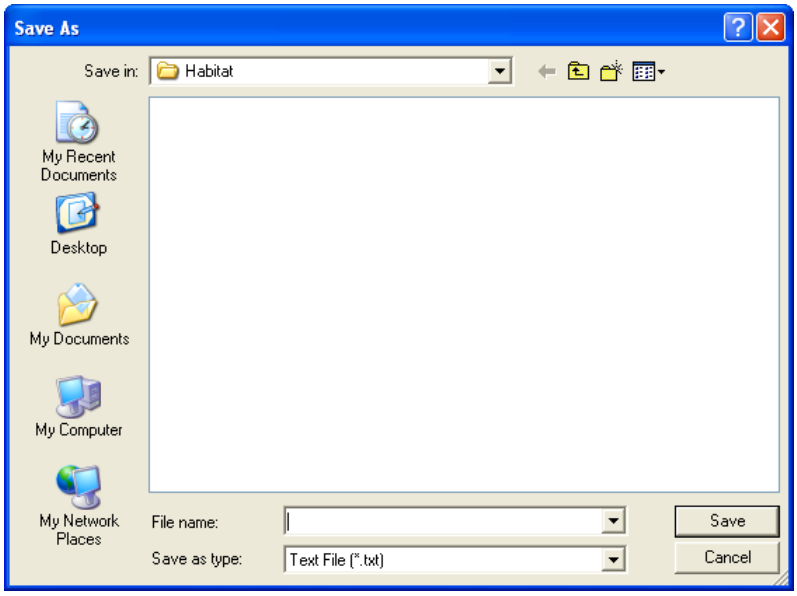

The scenario run log is saved for each individual execution of the scenario. The log contains information about the user running the scenario, the computer used to run the scenario, and the elements involved in the scenario. Each element run contributes a segment to the run log, including details about input and output files, and what actions the element took.

An example scenario log viewer is shown below.




*Exporting Scenario Run Log*

The scenario run log may be exported, which is useful for archiving scenario results. Exporting the log is done by clicking the "Export to File" button in the scenario log viewer (above). A Save As dialog box will appear, as shown below, allowing the user to select the output location and provide a file name for the log.




The output file will be textual (ASCII text), and will contain exactly what the scenario log viewer shows.

## Importing and Exporting Scenarios

Entire scenarios can be imported and exported using the DSS. Exporting a scenario is accomplished by first opening the scenario, then selecting the **File** menu, and then choosing **Export Scenario**. The user will be asked where to save the file.

The output file will be a proprietary binary format, which is unreadable by anything except the Model Manager / Scenario Builder.

Importing a scenario is accomplished in much the same way – start with an empty scenario builder screen, or with a scenario loaded. Click on the **File** menu and select **Import Scenario**. Select the file to be imported, and click Open.

The scenario will immediately open in the scenario builder, under a separate tab if another scenario is already open. The scenario will also be added to the database of scenarios automatically. If a scenario with the same name as the scenario being imported already exists in the database a warning message will appear and the scenario will not be imported.

An exported scenario file may be given to another user who has the Scenario Builder installed. As long as that user has all of the model elements that exist in the exported scenario and any input data that the model elements of that scenario require, the user can run the scenario.

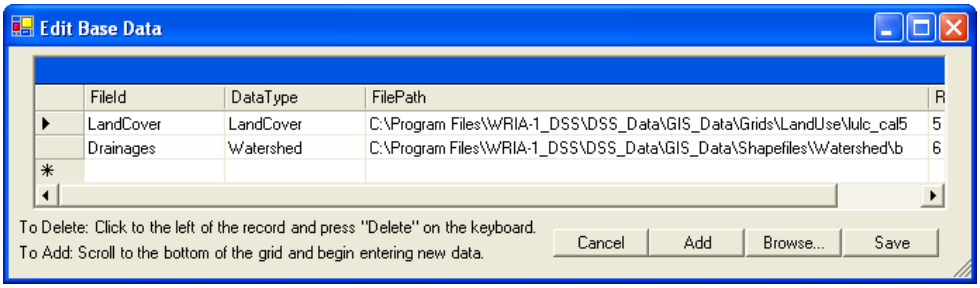
If a user imports a scenario and doesn't have one of the scenario elements, the missing element will show up in the Model Manager as a question mark in the scenario window.

It is useful to export a scenario run log as well, and keep it with the exported scenario– this is a useful way to tell what data the scenario requires. Each model element will list the data files it accesses as a part of the scenario log. (Custom elements not made by USU may not follow this guideline.)

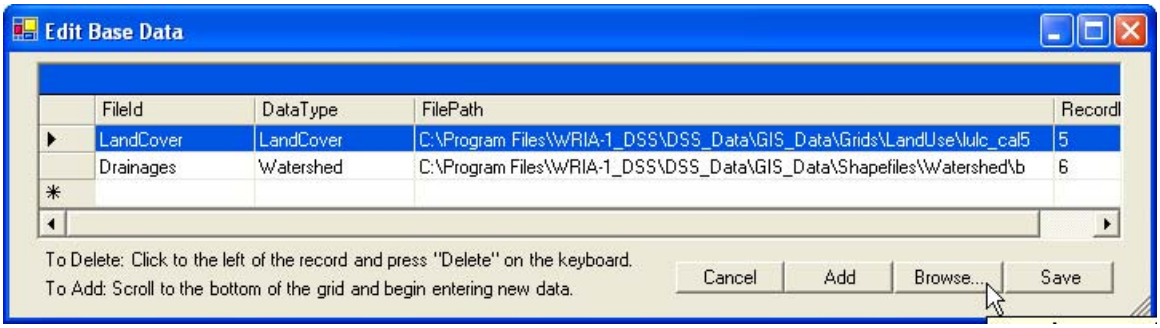
## Editing the Base Data

Part of the Scenario Builder database is a table listing all of the base data available to model elements. This table contains the data type, name and location.

This data may be edited by clicking on the **File** menu, then selecting **Edit Base Data**. The following screen will appear.




To change the location of an existing data item, click on the row to highlight it, click Browse... as shown below.



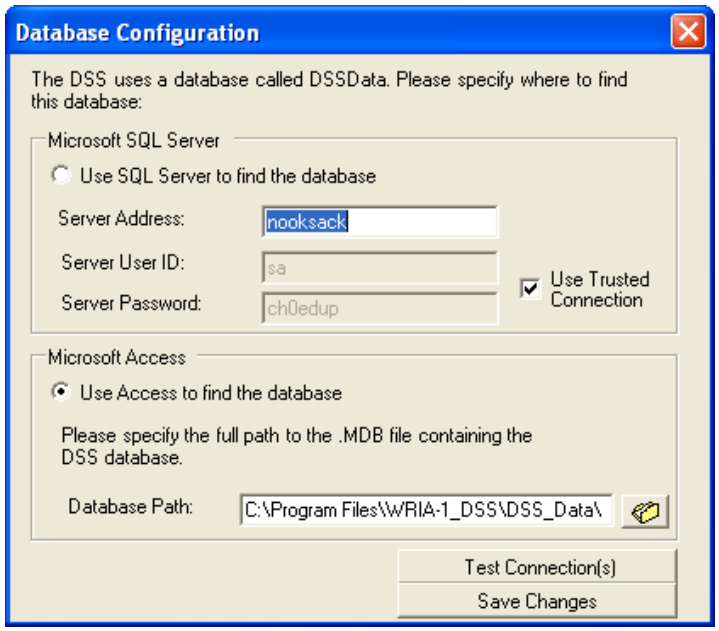

Navigate to the desired location and press Open. The new location will appear in the table on the screen. The FileID (data name) or DataType can be edited by clicking in the field and directly editing.

New data items can be added by clicking the Add button. To delete a data item, click in the grey area to the left of the record, highlighting the row, then press the Delete key on your keyboard to delete it.

None of these changes will be saved until the Save button is pressed, at which time all changes that have been made will be saved permanently.

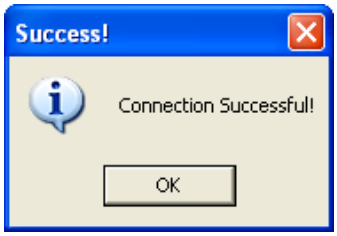
## Configuring the DSS Database

The Scenario Builder uses an internal database to store scenarios and base data information. The location of the database that the Scenario Builder is using may be changed by clicking on the **File** menu, then selecting **Database Configuration**. The screen will appear as shown below.




The database can be either an SQL Server database or a Microsoft Access database. Enter the server information if it is an SQL Server, or click the folder icon to browse for the Access database.

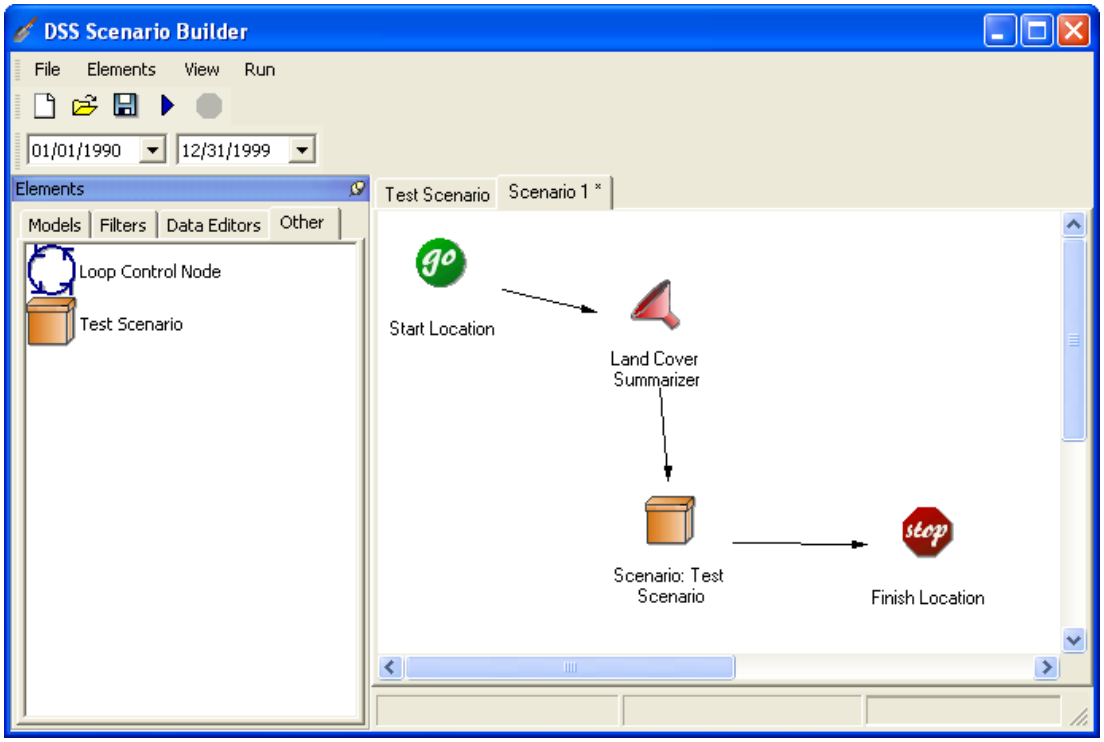
When you have filled in the proper information, click the Test Connection(s) button. You will see the message below.




Press OK and then Save Changes to continue.

## Placing a Scenario Inside of another Scenario

An entire scenario can placed within another scenario. This is best illustrated by example, as below:

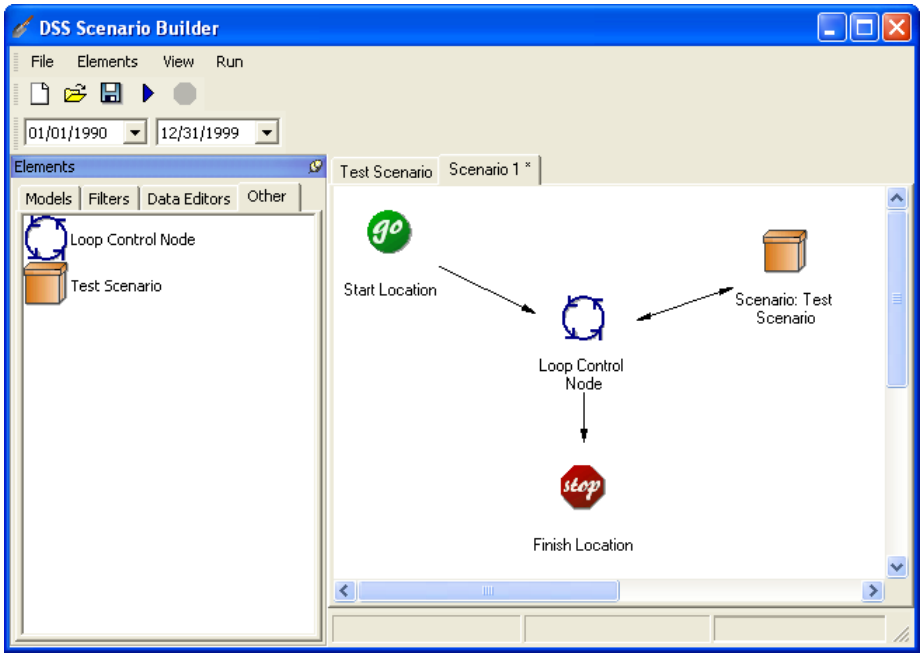



When the scenario shown above runs, the Land Cover Summarizer will run first, then the scenario "Test Scenario" will run, starting with the "Go" icon and ending at the "Stop" icon within that scenario. The scenario will then continue to the "Stop" icon in the window above. Additional elements could be placed between the "Scenario: Test Scenario" icon and the "Stop" icon if desired.

This may be useful to encapsulate often-used functions.

## The Loop Control Node

Another advanced scenario element is the loop control node. This facilitates running an element multiple times. This is also best illustrated by example:

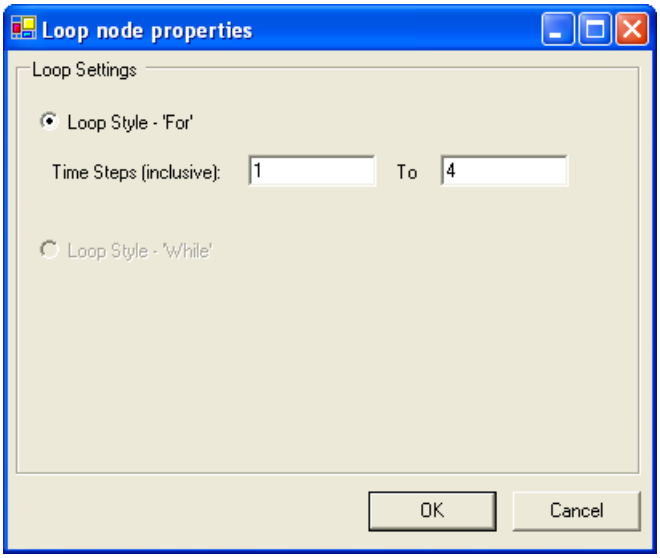



In the scenario shown above, the execution flows from "Go" to "Loop Control Node". The loop control node has been configured to run four times. Since the loop control node is connected to the node "Scenario: Test Scenario", it will be run four times before execution flows to "Stop".

The Loop Control Node must have a connection leading to a valid scenario element (not "Stop"), and a connection coming back into it. In this example, there is a line between "Loop Control Node" and "Scenario: Test Scenario" which has an arrowhead on both ends: this line is serving both roles.

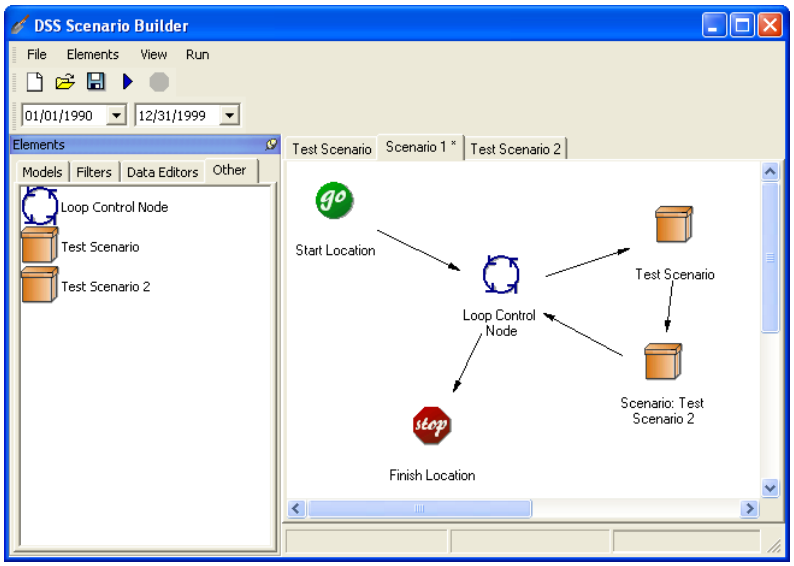
The Loop Control Node should then have a second connection leaving it, which in this example leads to "Stop".

The Loop Control Node is configured like any other element, simply double-click it. Double-clicking the Loop Control Node will produce the window below.




As this window shows, the loop control node will run for 4 time steps.

Another example is given below:




The Loop Control Node here has been set to run twice; so, execution will flow thusly:

1. (go icon)
2. (loop control node iteration 1)
3. Scenario: Scenario 1
4. Scenario: Scenario 2
5. (loop control node iteration 2)
6. Scenario: Scenario 1
7. Scenario: Scenario 2
8. (loop control node finished)
9. (stop icon)

## Custom Scenario Elements

The Scenario Builder has been put together with extensibility in mind. New model elements can easily be created using any programming language capable of implementing an interface from a DLL.

Details about constructing your own scenario elements can be found in the section entitled *Model Manager Technical Documentation* and *Programmer Tutorial: How to Create a DSS Model*.

# Land Cover Change Scenario Element

The Land Cover Change Element allows the DSS user to change the land cover of a land cover grid, and create a new summary of land cover within the changed land cover grid. The Land Cover Change Element consists of the Change Land Cover element and the Land Cover Summarizer element.

## Add Land Cover Change Element to DSS Scenario

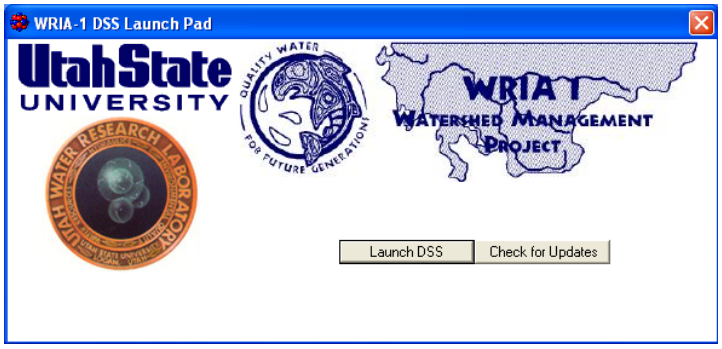
The Land Cover Change Element consists of the Change Land Cover element and the Land Cover Summarizer element. These elements must both be added to the same DSS Scenario.

### *Add Change Land Cover to DSS Scenario*

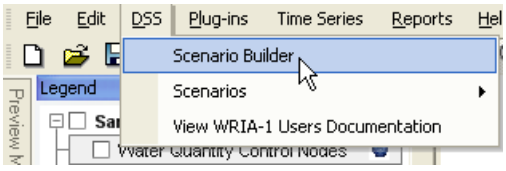
To add the Change Land Cover Element to a DSS scenario layout, first open the DSS Scenario Builder.

Double-click the WRIA-1 DSS icon

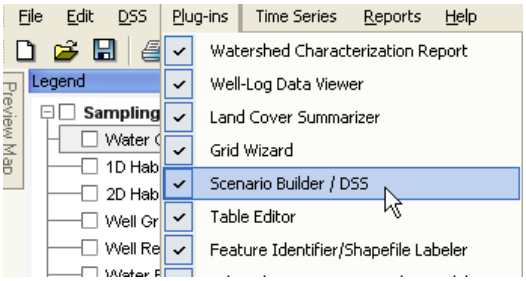
on the desktop (the icon with a red and blue molecule). This will open the WRIA-1 DSS Launch Pad window as shown below. Press **Launch DSS** to begin.




From the **DSS** menu, select the **Scenario Builder** option to start the Model Manager.

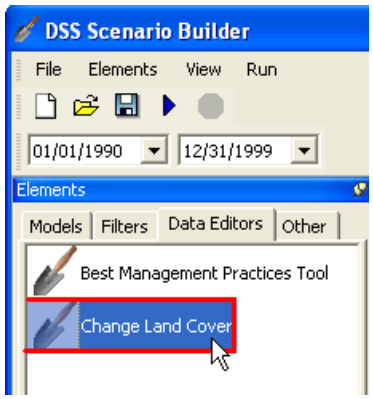


If this menu does not appear, select the **Plug-ins** menu instead. You will see an option labeled **Scenario Builder / DSS**. If there is no check mark next to this item, click the item as shown below. If the **Scenario Builder / DSS** option does not appear in the Plug-ins list, you do not have the Model Manager installed.

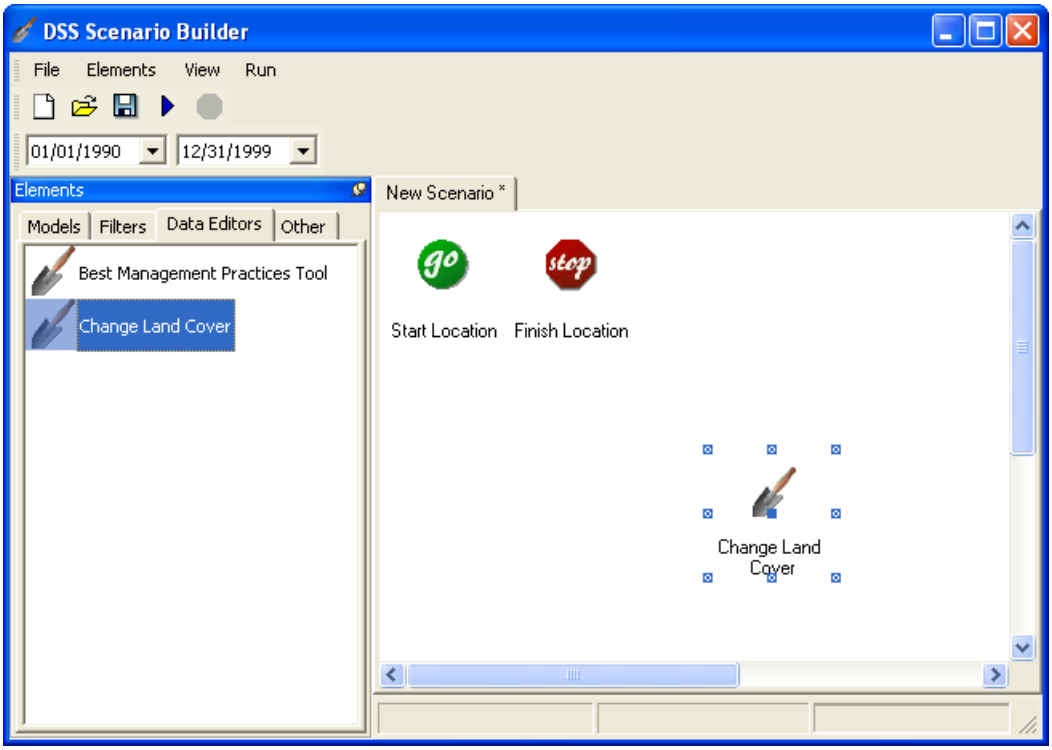



Once the DSS Scenario Builder window opens, begin a new scenario layout by pressing , or by selecting the **File** menu and choosing **New**.

Next, click on the Data Editors tab in the Elements window and then double click on the Change Land Cover element highlighted in red below.

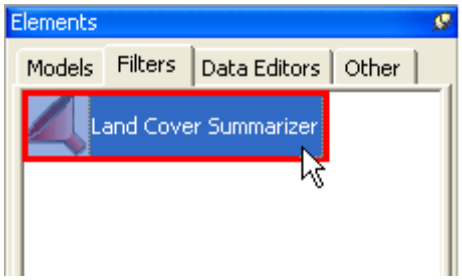



The Change Land Cover element will then be added to the scenario layout.

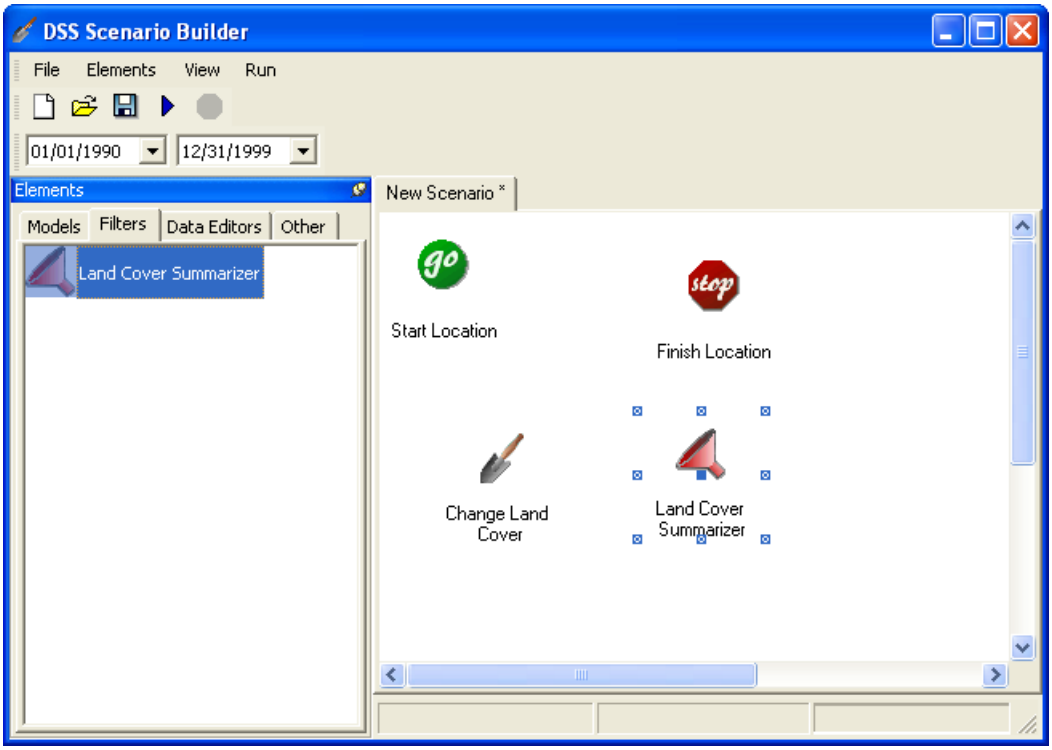



***Add Land Cover Summarizer to DSS Scenario***

Click on the Filters tab in the Elements window in the Scenario Builder and then double click on the Land Cover Summarizer element highlighted in red below.




The Land Cover Summarizer element will then be added to the scenario layout.




To prepare the Land Cover Change Scenario Element to be run in the Scenario Builder, you must connect the elements to the Start Location and the Finish Location nodes in the scenario layout.

First, click on the Start Location node. Then, click and hold the mouse on the box in the center of the node's icon. While still pressing the mouse button, drag the mouse to the Change Land Cover node in the scenario layout.

Next, click and hold the mouse button down on the box in the center of the Change Land Cover node, dragging the mouse to the Land Cover Summarizer node.

Finally, click and hold the mouse button down on the box in the center of the Land Cover Summarizer node, dragging the mouse to the Finish Location node.

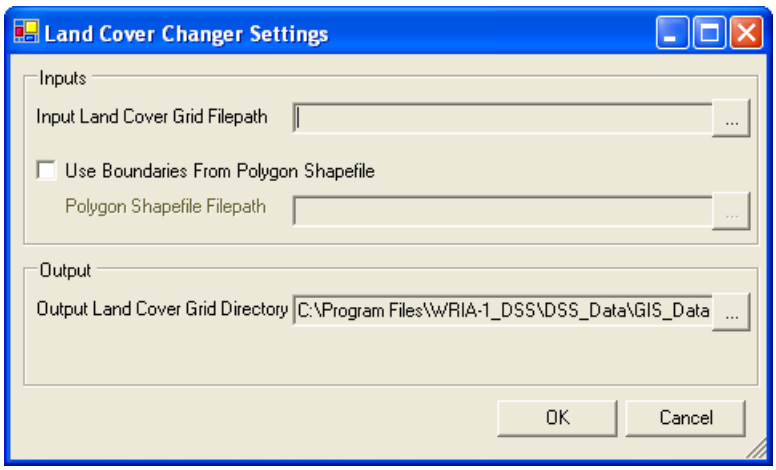
The Land Cover Changer Scenario is now ready to be run in the DSS Scenario Builder.

## Edit Land Cover Change Scenario Element Settings

The Land Cover Change Scenario is made up of the Change Land Cover element and the Land Cover Summarizer element. The Land Cover Change Scenario's settings are changed by editing the settings of these two individual elements.

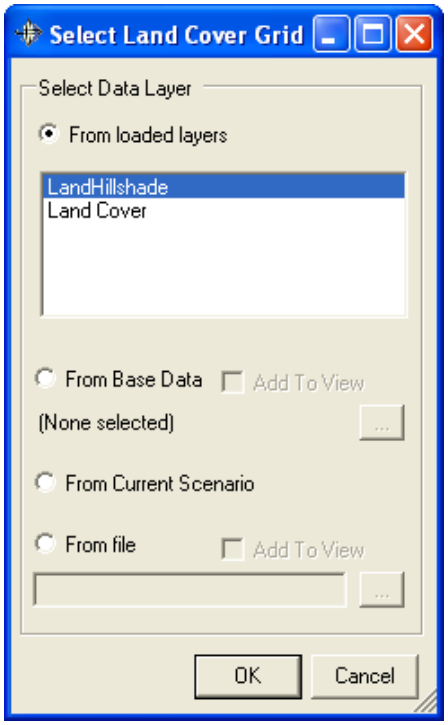
To edit the Change Land Cover element's settings, double click on the Change Land Cover node in the scenario layout.

The Change Land Cover Changer Settings form will be displayed as seen below.




***Select Input Land Cover Grid***

To select a land cover grid click the ... button to the right of Input Land Cover Grid Filepath in the screen shown above. The Select Land Cover Grid form will then be displayed as shown below.




To select a grid from the layers loaded in MapWindow's view, click the From loaded layers radio button, and then select the name of the grid layer in the layer list.

To select a grid from the DSS base data click the From Base Data radio button, then click the ... button. A base data form will be displayed with "LandCover" shown. You may add this grid to the current view by checking the box next to Add to View.

Add To View Click the OK button.

To select a grid from the current scenario, click the From Current Scenario radio button.

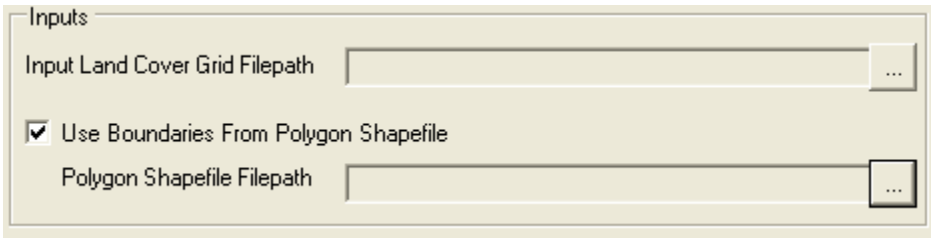
To select a grid from file click the From file radio button, then click the ... button. An open file dialog will be displayed. Browse to a land cover grid and click open. You may add this grid to the current view by checking the box next to Add to View.

Add To View Click the OK button to continue.

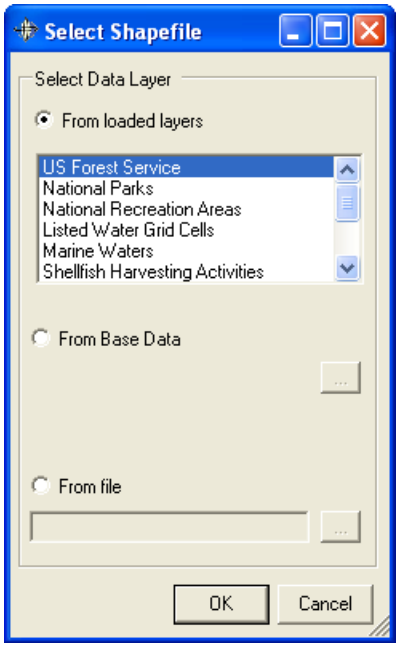
To save the land cover grid that has been selected click the OK button. To discard the land cover grid that has been selected click the Cancel button.

***Select Polygon Shapefile***

In order to use boundaries from polygon shapefiles, you must first check the box next Use Boundaries From Polygon Shapefile as shown below.




This will enable you to click the ... button to the right of Polygon Shapefile Filepath. The Select Shapefile form will then be displayed.




To select a shapefile loaded in MapWindow's view click the From loaded layers radio button, then select the name of the shapefile layer in the layer list.

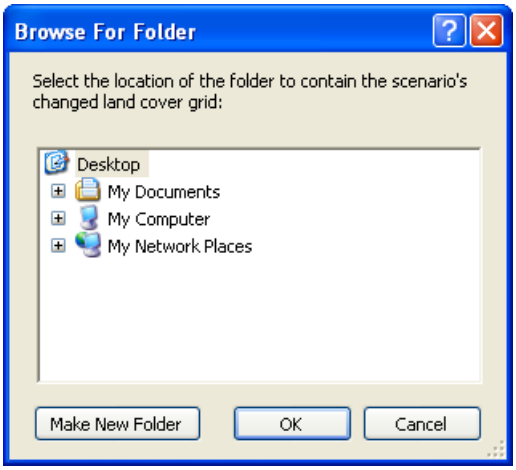
To select a shapefile from the DSS base data click the From Base Data radio button, then click the ... button. A base data form will be displayed with Drainages shown. Click the OK button.

To select a shapefile from file click the From file radio button, then click the ... button. An open file dialog will be displayed. Browse to a polygon shapefile and click open.

To save the polygon shapefile that has been selected click the OK button. To discard the polygon shapefile that has been selected click the Cancel button.

### *Select Output Land Cover Grid Directory*

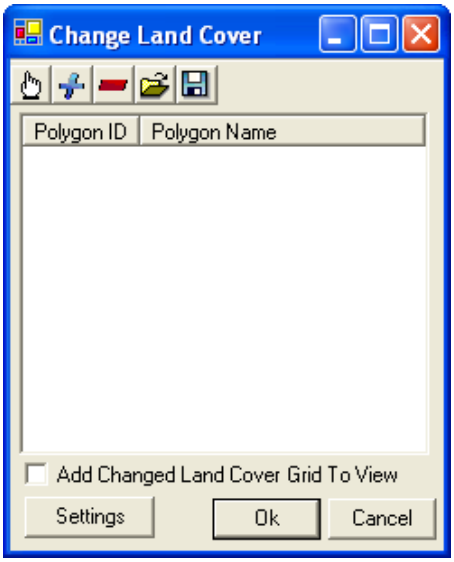
To select a directory to contain the changed land cover grid created by the Change Land Cover element, click the ... button to the right of Output Land Cover Grid Directory.




Browse to the directory to contain the changed land cover grid, and then click the OK button. Click the Cancel button to discard the directory selected.

### *Select Land Cover Change Polygons*

Once you have finished with the Change Land Cover Changer Settings, the Change Land Cover form will appear. This form allows the DSS user to select the changes in land cover for the specified input land cover grid.



*Add Land Cover Change Polygon*

To create a change in the land cover grid specified in the Change Land Cover Settings form, you must create a polygon containing specific changes in land cover.

*Add Shape from Polygon Shapefile*

To select a shape in the polygon shapefile specified in the settings click the button. The Change Land Cover form will be minimized, revealing the MapWindow view containing the land cover grid and polygon shapefile. Remember, you must have selected a polygon shapefile to use as the boundary in the previous step. If you have not, you will get an error message saying that you must select a polygon shapefile for the Land Cover Changer. Once you have done this you may continue.

The polygon shapefile will be the selected layer in MapWindow's Legend. To select a shape in the polygon shapefile, move the cursor over the desired shape and click on the map. The Select New Land Cover Types form will then be displayed allowing the user to select a new land cover type for old land cover types.

*Add Custom Drawn Polygon*

To draw a polygon on MapWindow's view click the button. The Change Land Cover form will be minimized, revealing the MapWindow view containing the land cover grid. To draw a shape in MapWindow's view, move the cursor over the desired starting point in the polygon and click on the map with the left mouse button. Repeat this process for all of the points desired in the polygon. If the user desires to remove the last point drawn, they can do so by clicking the right mouse button. When all of the polygon points have been added, the user can finish the polygon by moving the cursor over the start point until the cursor appears, and then clicking the left mouse button. The Select New Land Cover Types form will then be displayed allowing the user to select a new land cover type for old land cover types.

*Select New Land Cover Types*

The Select New Land Cover Types form is displayed when a new polygon is created, or when a polygon in the polygon list on the Change Land Cover form is double clicked. It allows the DSS user to select a new land cover type for an old land cover type in the polygon.

Select New Land Cover Types
Old Land Cover
New Land Cover
11 - Open Water
12 - Perennial Ice/Snow
21 - Low Intensity Residential
22 - High Intensity Residential
23 - Commercial/Industrial/Tran...
31 - Bare Rock/Sand/Clay
32 - Quarries/Strip Mines, Grave...
33 - Transitional
41 - Deciduous Forest
42 - Evergreen Forest
43 - Mixed Forest
51 - Shrubland
11 - Open Water
12 - Perennial Ice/Snow
21 - Low Intensity Residential
22 - High Intensity Residential
23 - Commercial/Industrial/Transp
31 - Bare Rock/Sand/Clay
32 - Quarries/Strip Mines, Gravel
33 - Transitional
41 - Deciduous Forest
42 - Evergreen Forest
43 - Mixed Forest
51 - Shrubland
*Right click or Double click on a Land Cover row to choose the new Land Cover type for the polygon.
View Only Land Cover Inside Polygon
OK
Cancel

To select a new land cover type for an old land cover type, click a row in the land cover types list. A menu will be displayed with the possible choices of new land cover type for that row. The current choice for the new land cover type will have a check mark beside it. Click the land cover that is the desired new land cover type and the new land cover type chosen will be displayed in the land cover types list for the appropriate row.

To limit the old land cover types listed in the land cover types list to the land cover found within the polygon being edited, click the View Only Land Cover Inside Polygon button. The View Only Land Cover Inside Polygon button text will change to be View All Land Cover. By clicking on View All Land Cover, the land cover types list will once again display all of the land cover types.

To accept the new land cover types selected click the OK button. To discard the new land cover types selected click the Cancel button.

*Remove Land Cover Change Polygon*

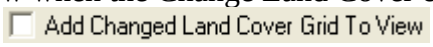
To remove a polygon from the polygon list, click on the polygon name in the list, and then click the button. The polygon will then be removed from the polygon list.

*Open/Save Land Cover Change Polygons List*

To open a saved list of polygons, click the button. An open file dialog will then be displayed, allowing the DSS user to browse to an xml file containing the saved polygons.

To save the polygons currently listed in the polygon list, click the button. A save file dialog will then be displayed, allowing the DSS user to select a location to save the polygons data in xml format.

*Add Changed Land Cover Grid to MapWindow View*

To add the grid containing the land cover changes selected for the polygons to MapWindow's view when the Change Land Cover element is run in the DSS Scenario Builder, check the 

box at the bottom of the Change Land Cover form.

*Edit Change Land Cover Settings*

To edit the settings for the Change Land Cover element, the DSS user may click on the

button. The Land Cover Changer Settings will then be displayed. These options are explained in the previous section titled Edit Land Cover Change Scenario Element Settings.

*Accept/Discard Land Cover Change Polygons*

To save the land cover changes selected in the Change Land Cover form, click the

button. The form will then be closed and the land cover changes will be saved in the Change Land Cover element.

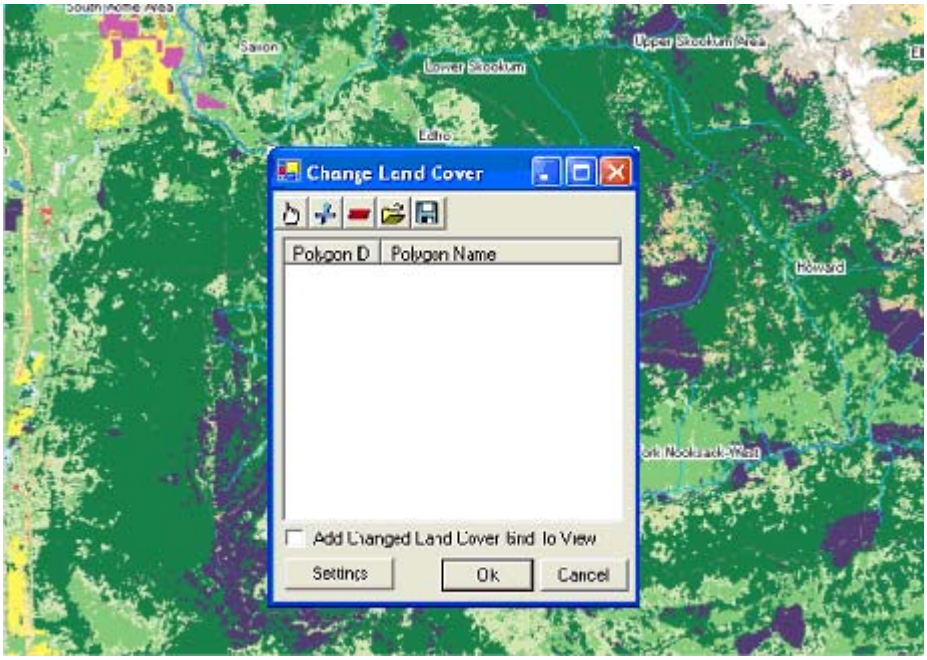
To cancel the land cover changes selected in the Change Land Cover form, click the

button. The form will then be closed and the land cover changes will be discarded.

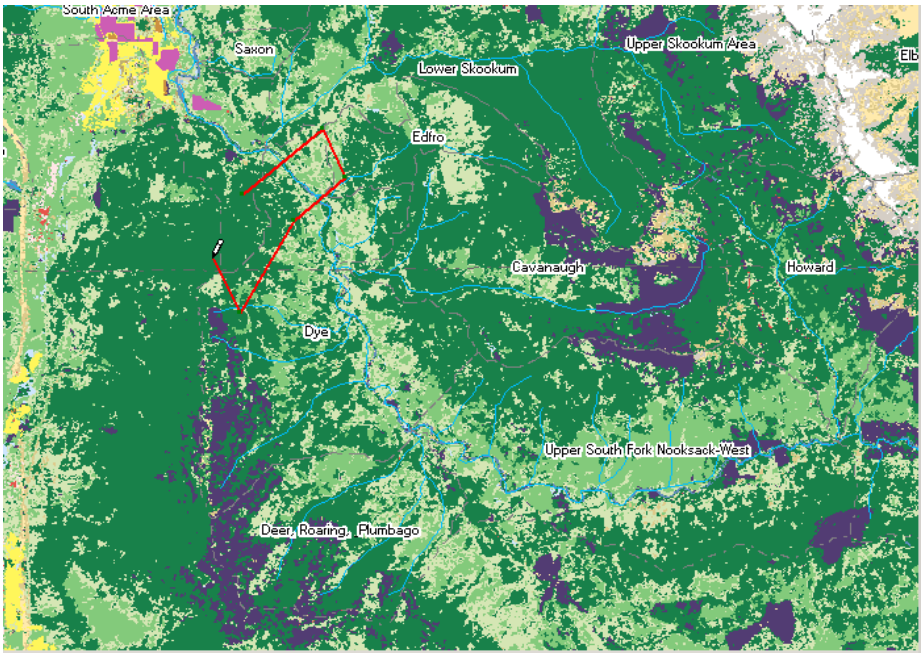
*Change Land Cover Example*

The following example shows how the DSS user can select polygons containing land cover changes.

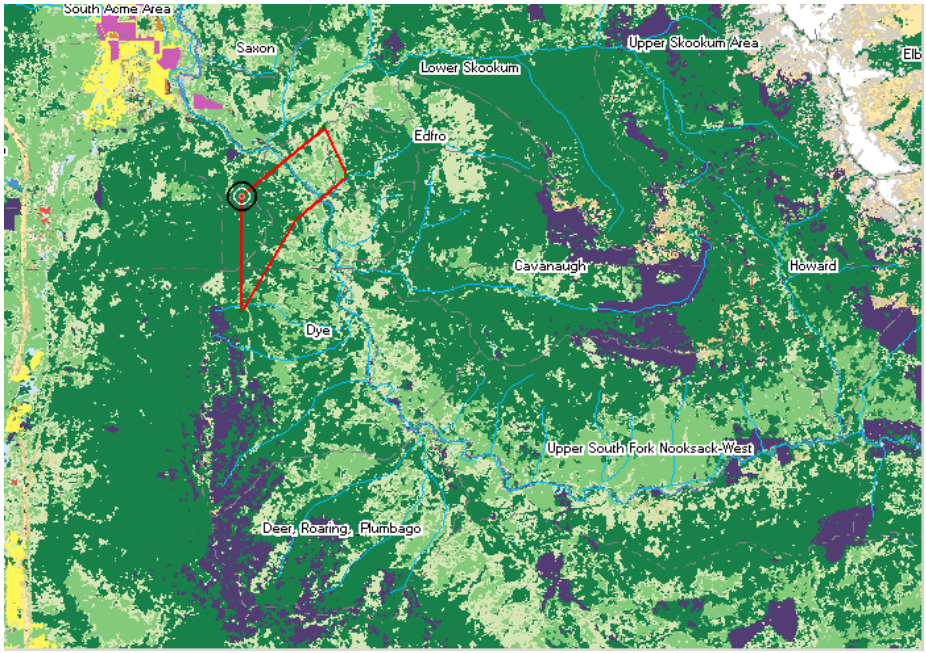
Once the Change Land Cover settings have been selected, the DSS user is ready to create polygons specifying the boundaries for land cover changes to be performed on the land cover grid. The figure below shows a Change Land Cover element with no polygons added yet.



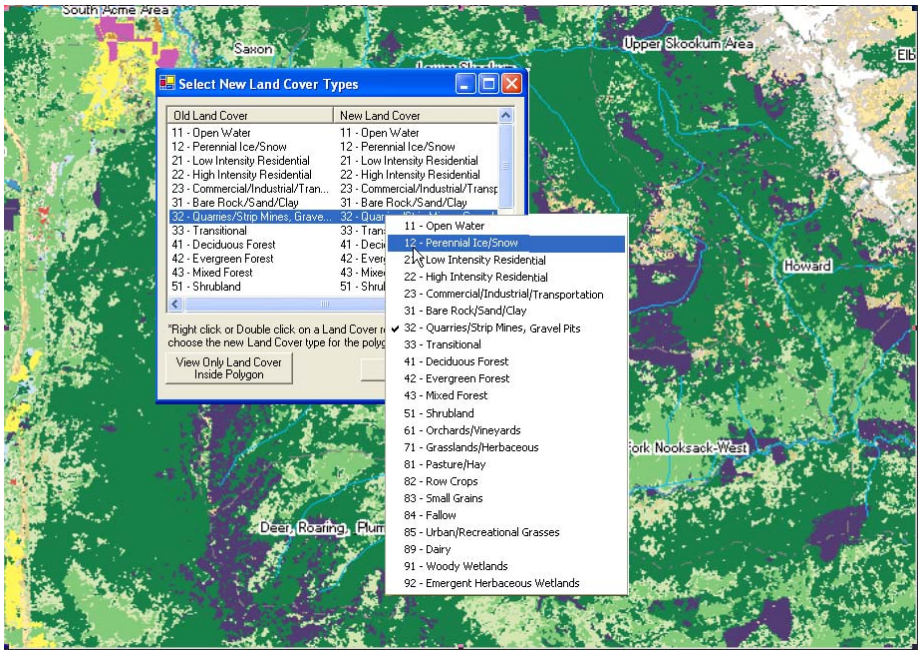

To draw a custom polygon on MapWindow's view, the user can click on the button. The Change Land Cover form will be minimized allowing the user to draw on MapWindow's view as shown below.




The user can add a point to the polygon being drawn by clicking on the map with the left mouse button, and can remove the last added point by clicking on the map with the right mouse button. When the user has added all of the desired points to the polygon being drawn, they can finish the polygon by clicking on the first point in the polygon with the left mouse button, as shown in the following figure.

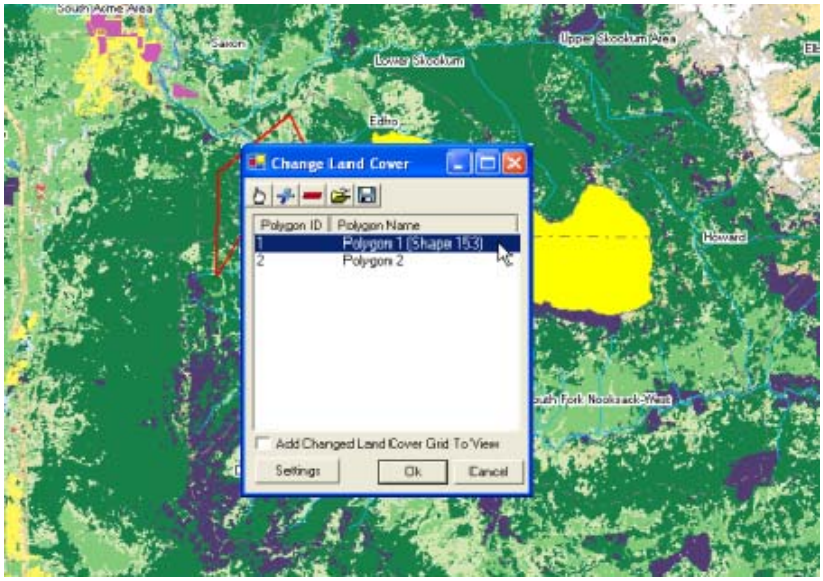



When the user has finished drawing the polygon, the Select New Land Cover Types form will be displayed as shown below. The user can then select new land cover types for the old land cover types by clicking on the desired row in the land cover list and selecting the new land cover type desired.


Select New Land Cover Types
Old Land Cover
New Land Cover
11 - Open Water
12 - Perennial Ice/Snow
21 - Low Intensity Residential
22 - High Intensity Residential
23 - Commercial/Industrial/Tran...
31 - Bare Rock/Sand/Clay
32 - Quarries/Strip Mines, Grave...
33 - Transitional
41 - Deciduous Forest
42 - Evergreen Forest
43 - Mixed Forest
51 - Shrubland
"Right click or Double click on a Land Cover r
choose the new Land Cover type for the polyg
View Only Land Cover Inside Polygon
11 - Open Water
12 - Perennial Ice/Snow
21 - Low Intensity Residential
22 - High Intensity Residential
23 - Commercial/Industrial/Transportation
31 - Bare Rock/Sand/Clay
32 - Quarries/Strip Mines, Gravel Pits
33 - Transitional
41 - Deciduous Forest
42 - Evergreen Forest
43 - Mixed Forest
51 - Shrubland
61 - Orchards/Vineyards
71 - Grasslands/Herbaceous
81 - Pasture/Hay
82 - Row Crops
83 - Small Grains
84 - Fallow
85 - Urban/Recreational Grasses
89 - Dairy
91 - Woody Wetlands
92 - Emergent Herbaceous Wetlands
Howard
Deer, Roaring, Plum

When the new land cover types have been selected, the user clicks the OK button, and is returned to the Change Land Cover form which lists all of the polygons added to the

Change Land Cover element in the polygon list. The next figure shows two polygons that have been added to the Change Land Cover element.

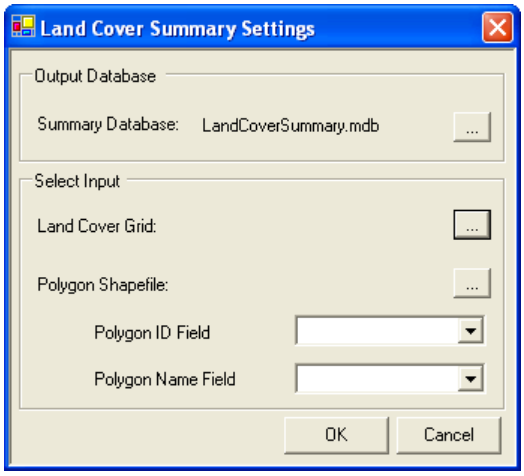



### *Edit Land Cover Summarizer Settings*

The Land Cover Summary Settings for allows the user to change the Land Cover Summarizer element settings.

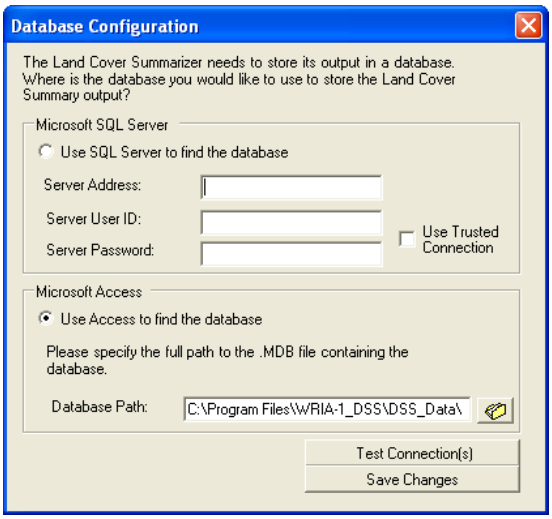
To edit the Land Cover Summarizer element's settings, double click on the Land Cover Summarizer node in the scenario layout.

The Land Cover Summarizer element settings form will be displayed as seen below.




*Select Output Land Cover Summary Database*

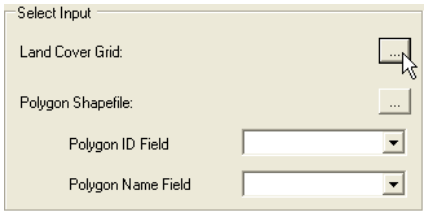
To select the Land Cover Summary Database for the Land Cover Summarizer output, click the ... button to the right of Summary Database:.


Database Configuration
The Land Cover Summarizer needs to store its output in a database. Where is the database you would like to use to store the Land Cover Summary output?
Microsoft SQL Server
Use SQL Server to find the database
Server Address:
Server User ID:
Server Password:
Use Trusted Connection
Microsoft Access
Use Access to find the database
Please specify the full path to the .MDB file containing the database.
Database Path:
C:\Program Files\WRIA-1_DSS\DSS_Data\
Test Connection(s)
Save Changes

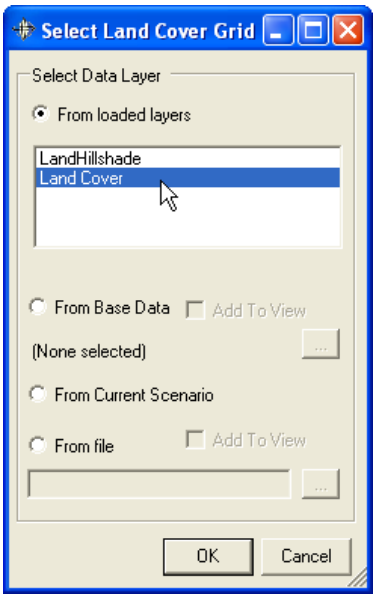
To browse to a Microsoft Access Land Cover Summary Database click the button. An open file dialog will be displayed. To save the selected database location, click the Save Changes button.

*Select Land Cover Grid*

To select a land cover grid click the ... button to the right of Land Cover Grid: in the Land Cover Summary Settings form.




The Select Land Cover Grid form will then be displayed.



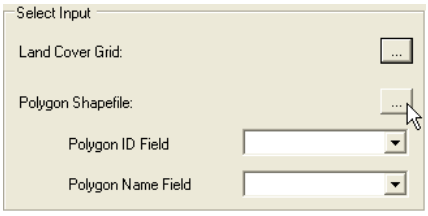

To select a grid loaded in MapWindow's view click the From loaded layers radio button, and then select the name of the grid layer in the layer list.

To select a grid from the DSS base data click the From Base Data radio button, then click the ... button. A base data form will be displayed with LandCover shown. Click the OK button.

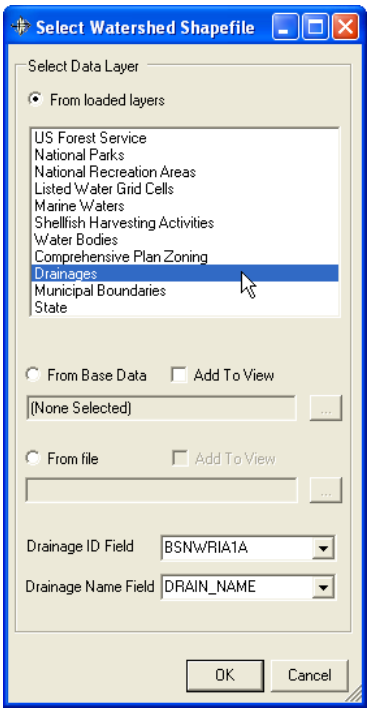
To save the land cover grid that has been selected click the OK button. To discard the land cover grid that has been selected click the Cancel button.

*Select Polygon Shapefile*

To select a polygon shapefile click the ... button to the right of Polygon Shapefile:.




The Select Shapefile form will then be displayed.




To select a shapefile loaded in MapWindow's view click the From loaded layers radio button, then select the name of the shapefile layer in the layer list.

To select a shapefile from the DSS base data click the From Base Data radio button, then click the ... button. A base data form will be displayed with Drainages shown. Click the OK button.

To select a shapefile from file click the From file radio button, then click the ... button. An open file dialog will be displayed. Browse to a polygon shapefile and click open.

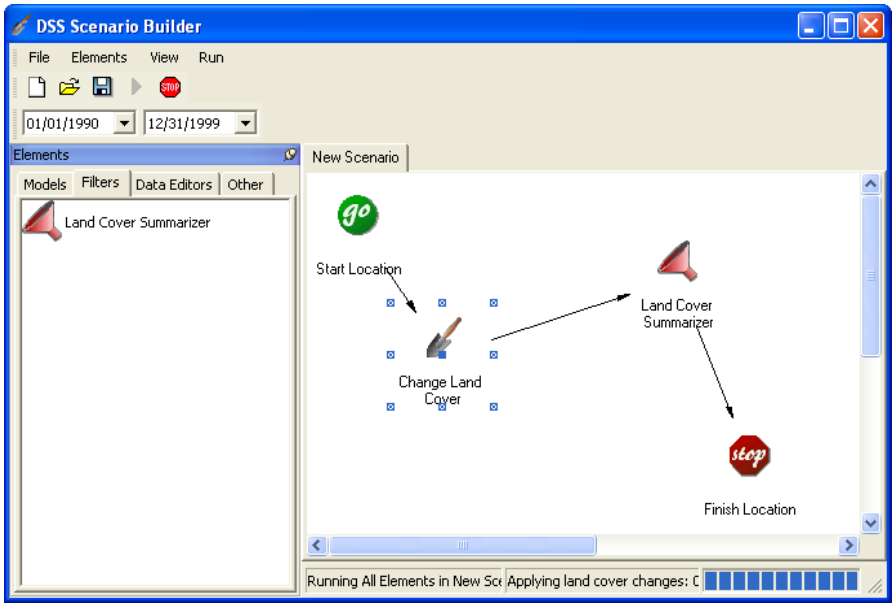
To save the polygon shapefile that has been selected click the OK button. To discard the polygon shapefile that has been selected click the Cancel button.

To select the shapefile field containing the ID for the polygons in the shapefile select a field name in the drop down list to the right of Polygon ID Field. To select the shapefile field

containing the name of the polygons in the shapefile select a field name in the drop down list to the right of Polygon Name Field.

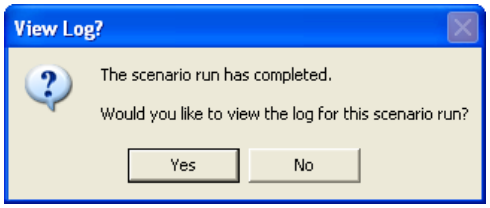
## Run Land Cover Change Scenario Element

Once all the elements have been configured, the Land Cover Change Scenario is ready to be run.




To run the scenario, click the button. If you would like to abort the scenario run, click the button to cancel the scenario run.

When the scenario has completed, the DSS user will be asked if they would like to view the scenario run log (a text file containing messages output by the elements contained in the scenario that has just been run).




To view the log for the scenario run, click the Yes button. To decline viewing the scenario log at this time click the No button.

# Water Quantity Model Interface

The Water Quantity Model Interface allows the DSS user to specify the input for the water quality model for the WRIA-1 Drainages. The water quantity model requires the following input information:

- Drainages Shapefile
- TopNet Input Directory
- Land Cover Summary Database
- Water Management Database

The drainage shapefile used should be bsnwria1_v7.shp. Only the drainage identifier (BSNWRIA1A) and drainage name (DRAIN_NAME) are used from this shapefile. Other inputs have been created using the drainage identifiers in this shapefile. Use of another shapefile would result in incorrect names being associated with drainage identifiers.

The TopNet input directory is the folder comprising input files required to run the water quantity model. Three such folders are provided, one each for historic, existing and full build out conditions. When the model is run this entire set of files is copied to the specified output working directory. The model is then run in the output working directory which serves as a record of the inputs and outputs for a specific model run scenario.

The land cover summary database contains the area of each land cover type summarized for each drainage. This is created by the land cover summarizer tool. The initial land cover summary database provided includes land covers for each scenario:

- Scenario 1 = Historic conditions
- Scenario 2 = Existing conditions
- Scenario 3 = Full build out conditions

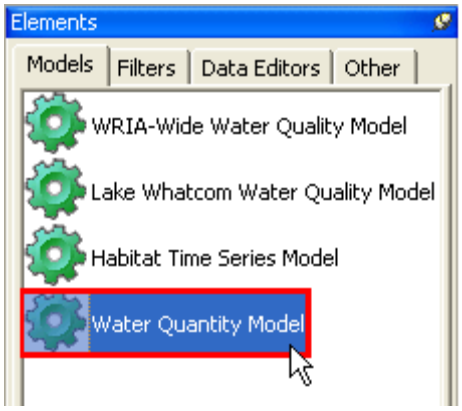
The water management database is a Microsoft Access database contains the information needed to populate the water quantity model water management input files. This database is changed when water management options are edited. Three initial water management databases are provided, one for each scenario.

The following sections describe the steps involved in using the water quantity model.

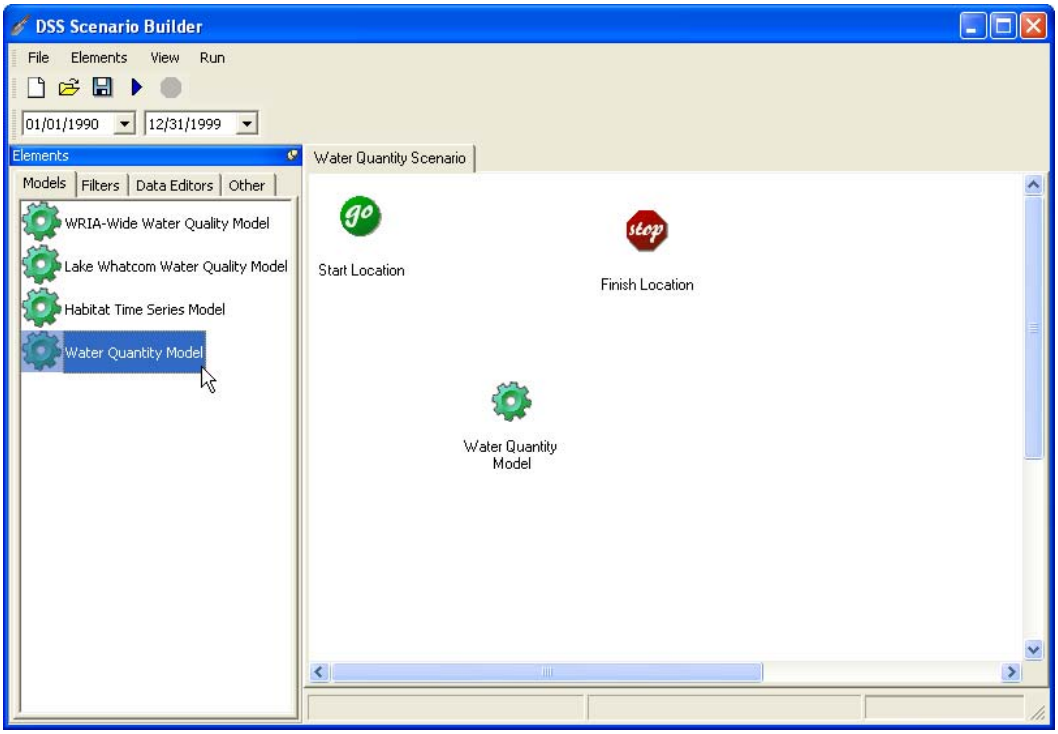
## Add Water Quantity Model to DSS Scenario

To add the Water Quantity Model to a DSS scenario layout, first open the DSS Scenario Builder by choosing the Scenario Builder option under the DSS menu. Create a new

scenario layout by clicking . Click on the Models tab in the Elements window of the Scenario Builder and double click on the Water Quantity Model highlighted in red below.

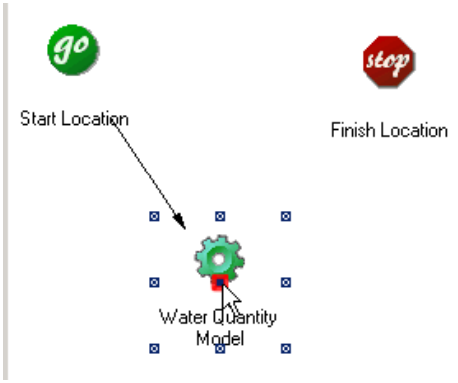



The Water Quantity Model will be added to the scenario layout.

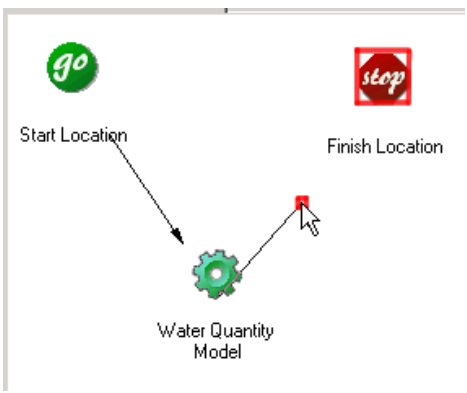



To prepare the Water Quantity Model to be run in the Scenario Builder, you must connect the model to the Start Location and the Finish Location nodes in the scenario layout.

First, click on the Start Location node. Then, click and hold the mouse on the box in the center of the node's icon. While still pressing the mouse button, drag the mouse to the Water Quantity Model node in the scenario layout.



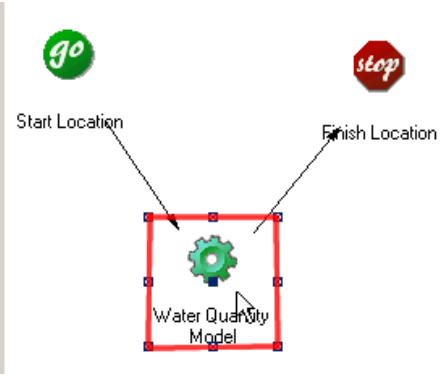

Repeat the previous step by again clicking the box in the center of the Water Quantity Model, dragging the mouse to the Finish Location node.



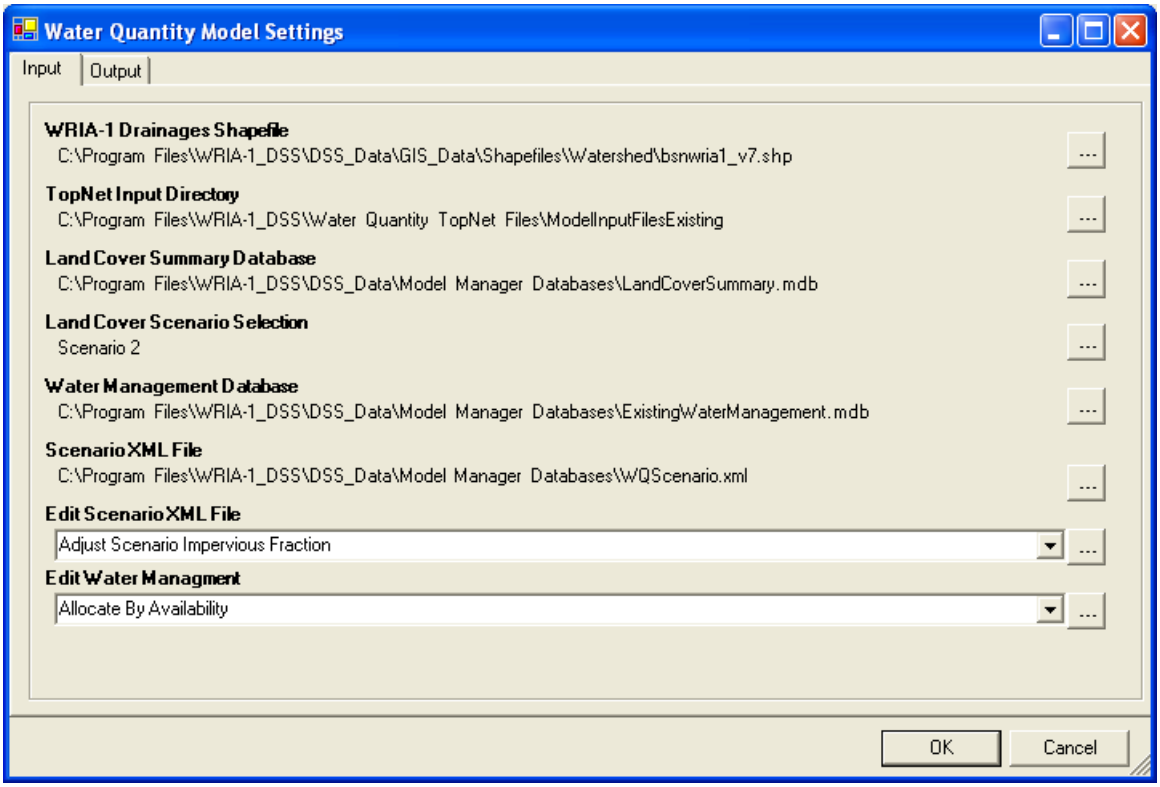

The Water Quantity Model is now ready to be run in the DSS Scenario Builder.

## Edit Water Quantity Model Settings

To edit the Water Quantity Model settings, double click on the Water Quantity Model node in the scenario layout.




The Water Quantity Model settings form will be displayed as seen below.



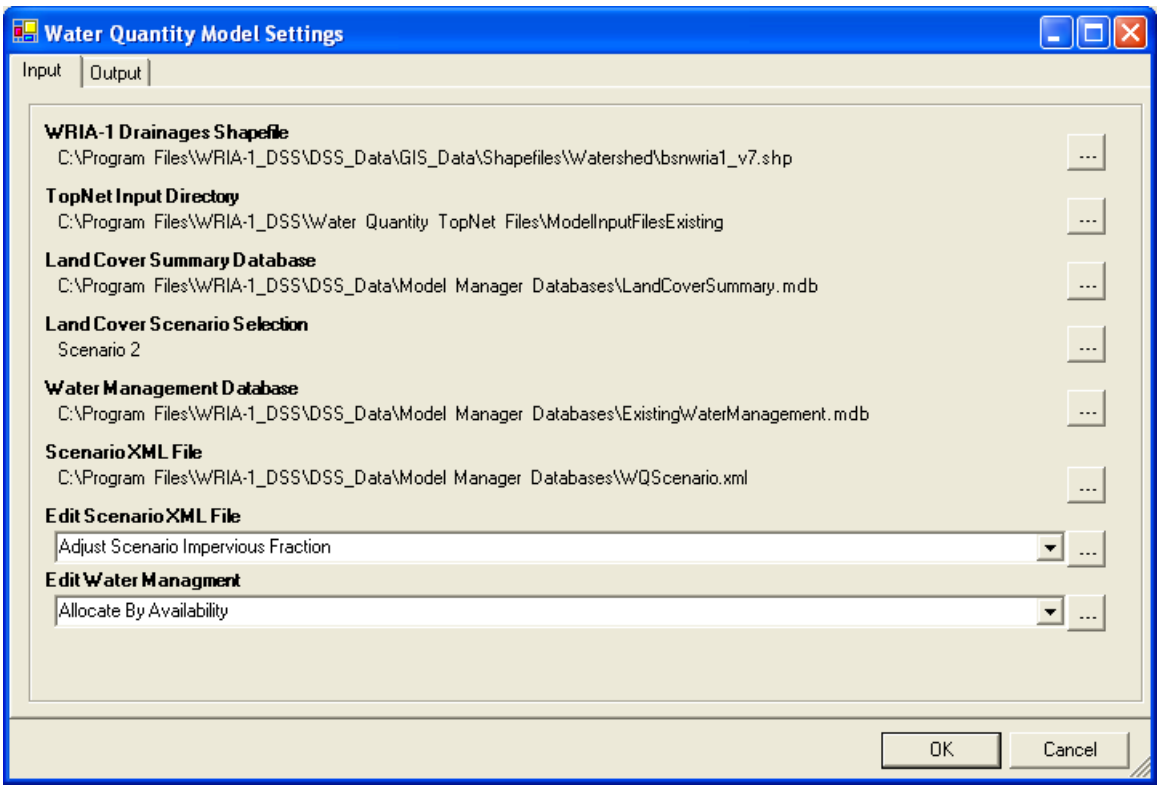

The water quantity model takes input from files in the TopNet input directory. There is one directory for each of the three model scenarios (Historic, Existing and Full Build Out conditions). When the model is run this entire set of files is copied to the specified output working directory. The model is then run in the output working directory which serves as a record of the inputs and outputs for a specific model run scenario. The Input tab gives

the capability for adjusting input information. The output tab specifies the locations of the output. The input directory is left unchanged.

To edit the input settings, click on the input tab. To edit the output settings, click on the output tab.

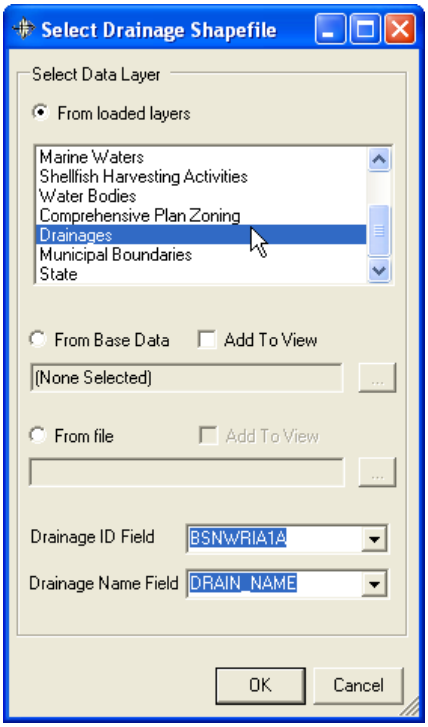
### *Edit Water Quantity Model Inputs*

The input tab of the Water Quantity Model Settings allows the DSS user to edit the inputs for the Water Quantity Model.




*Select WRIA-1 Drainages Shapefile*

To select the WRIA-1 Drainages Shapefile click the ... button next to WRIA-1 Drainages Shapefile. The Select Drainage Shapefile form will then be displayed.




To select a shapefile loaded in MapWindow's view click the From loaded layers radio button, then select the name of the shapefile layer in the layer list.

To select a shapefile from the DSS base data click the From Base Data radio button, then click the ... button. A base data form will be displayed with "Drainages" shown. Click the OK button.

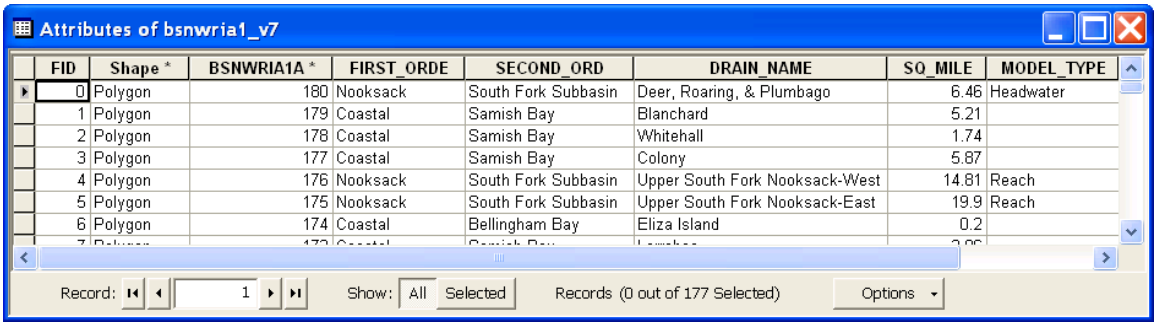
To select a shapefile from file click the From file radio button, then click the ... button. An open file dialog will be displayed. Browse to a polygon shapefile and click open.

To select the shapefile field to be used for the id, choose a field listed in the dropdown list to the right of Drainage ID Field. To select the shapefile field to be used for the name, choose a field listed in the dropdown list to the right of Drainage Name Field.

To save the polygon shapefile that has been selected click the OK button. To discard the polygon shapefile that has been selected click the Cancel button.

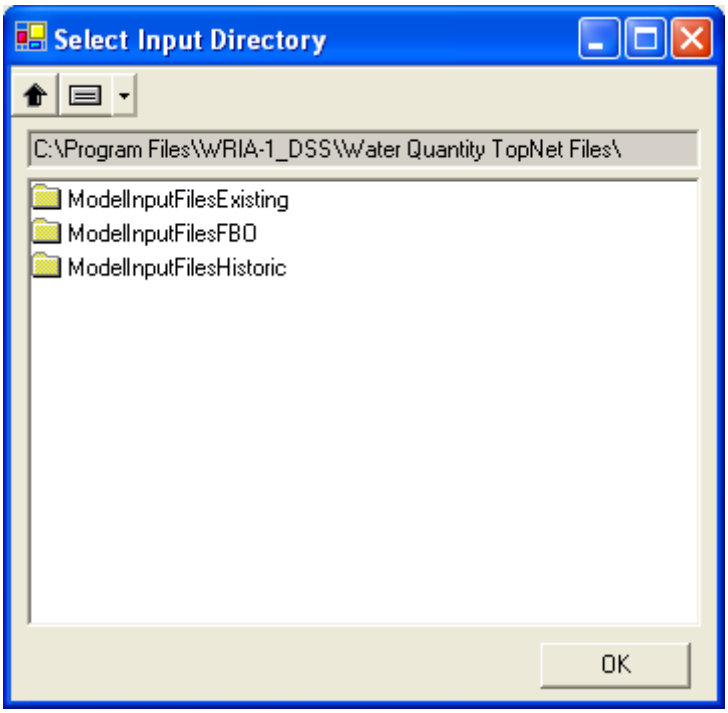
The drainage shapefile used should always be bsnwria1_v7.shp in the folder C:\program files\WRIA-1_DSS\DSS_Data\GIS_Data\Shapefiles\Watershed or equivalent location depending on your install location. Only the drainage identifier (BSNWRIA1A) and drainage name (DRAIN_NAME) are used from this shapefile. Other inputs have been created using the drainage identifiers in this shapefile. Use of another shapefile would result in incorrect names being associated with drainage identifiers. The model reads the

drainage identifier from the 3rd column and drainage name from the 6th column of the shapefile. A shapefile with this information in different columns will result in an error.


Attributes of bsnwria1_v7
FID | Shape * | BSNWRIA1A * | FIRST_ORDE | SECOND_ORD | DRAIN_NAME | SQ_MILE | MODEL_TYPE
0 | Polygon | 180 | Nooksack | South Fork Subbasin | Deer, Roaring, & Plumbago | 6.46 | Headwater
1 | Polygon | 179 | Coastal | Samish Bay | Blanchard | 5.21 |
2 | Polygon | 178 | Coastal | Samish Bay | Whitehall | 1.74 |
3 | Polygon | 177 | Coastal | Samish Bay | Colony | 5.87 |
4 | Polygon | 176 | Nooksack | South Fork Subbasin | Upper South Fork Nooksack-West | 14.81 | Reach
5 | Polygon | 175 | Nooksack | South Fork Subbasin | Upper South Fork Nooksack-East | 19.9 | Reach
6 | Polygon | 174 | Coastal | Bellingham Bay | Eliza Island | 0.2 |
Record: 1
Show: All Selected
Records (0 out of 177 Selected)
Options

*Select TopNet Input Directory*

To select an input directory for the TopNet model click the ... button to the right of **TopNet Input Directory**. The Select Directory form will then be displayed. The Select Input Directory form allows the DSS user to select a directory containing the input for the TopNet model. The default location for the TopNet Input Directory is "C:\Program Files\WRIA-1_DSS\Water Quantity TopNet Files\ModelInputFilesExisting".

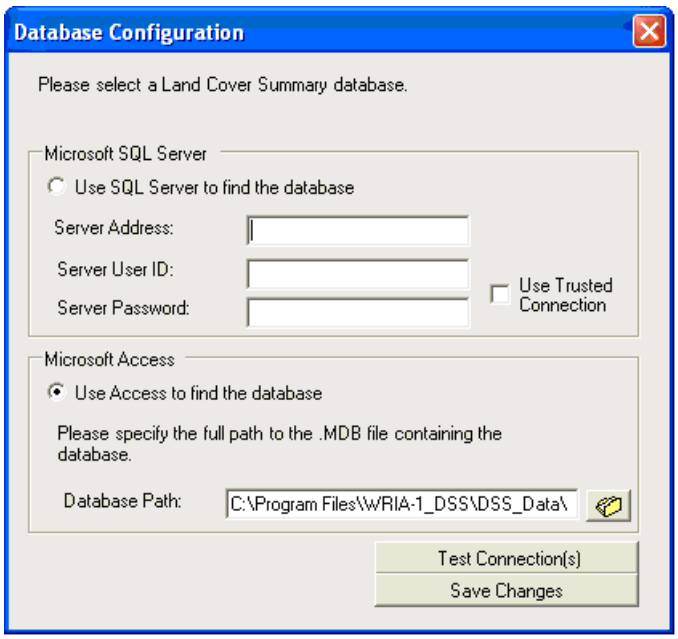



To move up a directory level click the button. To change the drive to browse for folders click the button. To select a folder as the desired directory, click on the folder's name. To open a folder double-click the folder's name.

To save the selected directory, click the OK button.

*Select Land Cover Database*

To select the Land Cover Summary Database, click the ... button to the right of Land Cover Database.

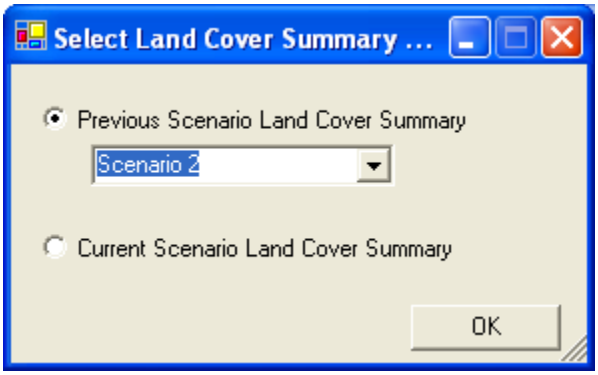



To browse to a Microsoft Access Land Cover Summary Database click the button. An open file dialog will be displayed. The default location for the Land Cover Summary Database is "C:\Program Files\WRIA-1_DSS\DSS_Data\Model Manager Databases\LandCoverSummary.mdb".

To save the selected database location, click the Save Changes button.

*Select Land Cover Scenario*

To select the scenario ID for the land cover summary to be used by the Water Quantity Model, click the ... button to the right of Land Cover Scenario Selection. The Select Land Cover Summary Scenario form will then be displayed.




To use a land cover summary from a previous scenario, click the Previous Scenario Land Cover Summary radio button, and then select a scenario from the drop down menu. To select a land cover summary that will be created when the current scenario layout is run, click the Current Scenario Land Cover Summary radio button. To save the scenario that has been selected click the OK button.

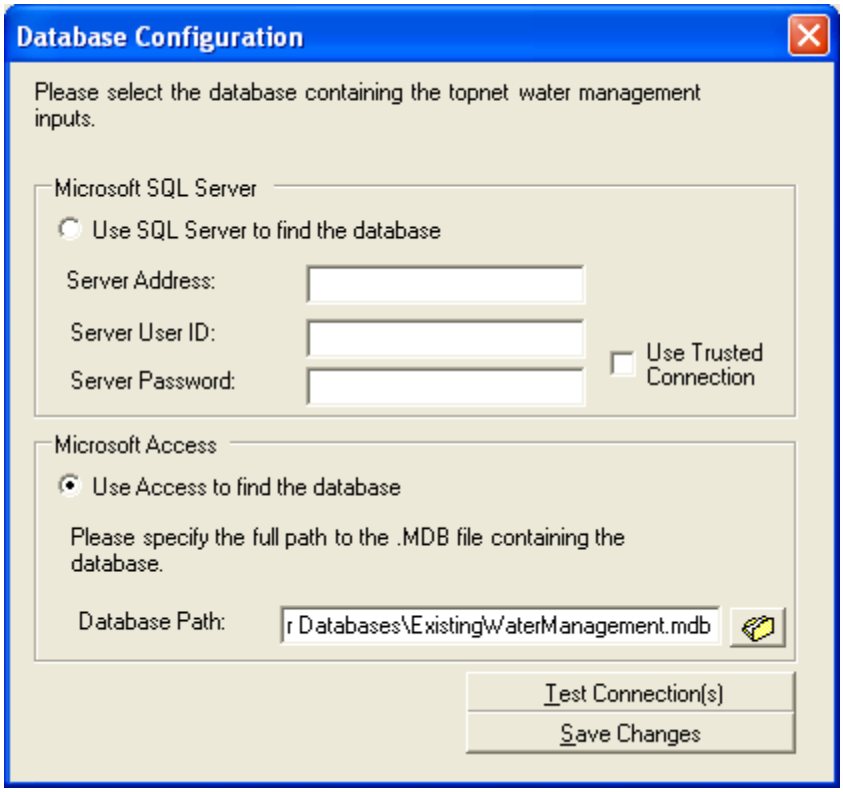
The initial land cover summary database provided includes land covers for each scenario:

- Scenario 1 = Historic conditions
- Scenario 2 = Existing conditions
- Scenario 3 = Full build out conditions

As additional land cover scenarios are created the user needs to keep track of what they are and which database they are in.

*Select Water Rights Database*

To select the Water Rights Database, click the ... button to the right of Water Rights Database.


Database Configuration
Please select the database containing the topnet water management inputs.
Microsoft SQL Server
Use SQL Server to find the database
Server Address:
Server User ID:
Server Password:
Use Trusted Connection
Microsoft Access
Use Access to find the database
Please specify the full path to the .MDB file containing the database.
Database Path:
r Databases\ExistingWaterManagement.mdb
Test Connection(s)
Save Changes

To browse to a Microsoft Access Water Management Database click the button. An open file dialog will be displayed. The default location for the Water Management Database is "C:\Program Files\WRIA-1_DSS\DSS_Data\Model Manager Databases\ExistingWaterManagement.mdb".

To save the selected database location, click the Save Changes button.

Three initial water management databases are provided, one for each scenario:

- HistoricWaterManagement.mdb
- ExistingWaterManagement.mdb
- FBOWaterManagement.mdb

*Select Scenario XML File*

The scenario XML file is used to store information on drainage area within each land cover class within each drainage and parameters pertaining to imperviois area and agricultural drainage which are editable. A default scenario XML file is provided and will be updated whenever the scenario is changed. However you may save and use and alternate XML file. To select an XML file, click the ... button to the right of Scenario XML File in the Water Quality Model Settings form. An open file dialog will be displayed, allowing you to browse to the desired scenario XML file. The default scenario

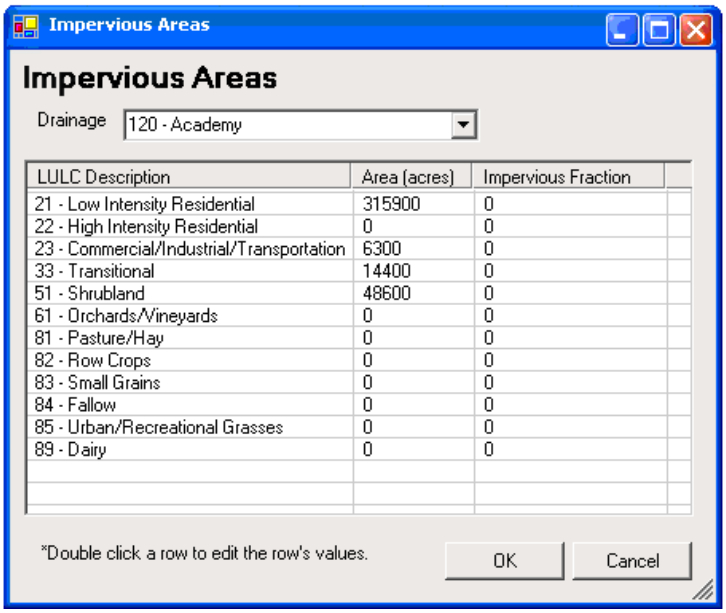
XML file is: C:\Program Files\WRIA-1_DSS\DSS_Data\Model Manager Databases\WQScenario.xml

*Edit Scenario XML File*

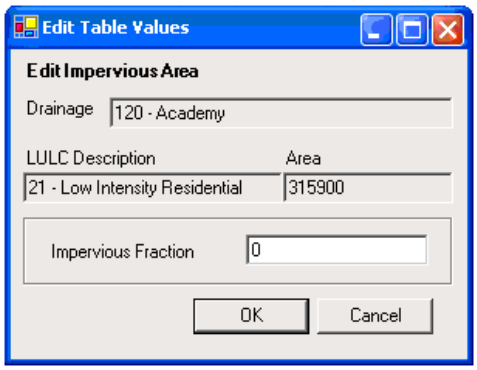
To edit the Impervious and Agricultural Drainage values for the scenario XML file, click the ... button to the right of Edit Scenario XML File. If Adjust Scenario Impervious Fraction is selected in the drop down menu, the Impervious Areas form will be displayed. If Adust Scenario Agricultural Drainage Values is selected in the drop down menu, the Agricultural Drainage form will be displayed.

*Edit Scenario XML File Impervious Areas Values*

The Impervious Areas form allows the DSS user to edit the Impervious values for a WRIA-1 Drainage. These values are then stored in the scenario XML file.


Impervious Areas
Impervious Areas
Drainage
120 - Academy
LULC Description
Area (acres)
Impervious Fraction
21 - Low Intensity Residential 315900 0
22 - High Intensity Residential 0 0
23 - Commercial/Industrial/Transportation 6300 0
33 - Transitional 14400 0
51 - Shrubland 48600 0
61 - Orchards/Vineyards 0 0
81 - Pasture/Hay 0 0
82 - Row Crops 0 0
83 - Small Grains 0 0
84 - Fallow 0 0
85 - Urban/Recreational Grasses 0 0
89 - Dairy 0 0
*Double click a row to edit the row's values.
OK
Cancel

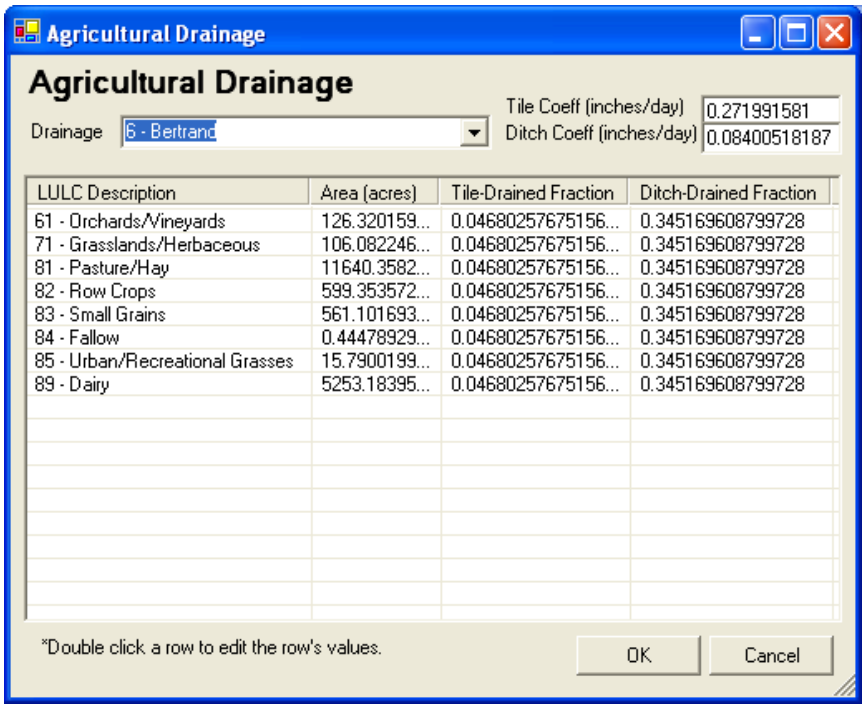
To select a WRIA-1 Drainage, click the drop down menu to the right of Drainage. To edit an impervious fraction for a land cover type, double click on the row in the land cover list. The Edit Table Values form will then be displayed.



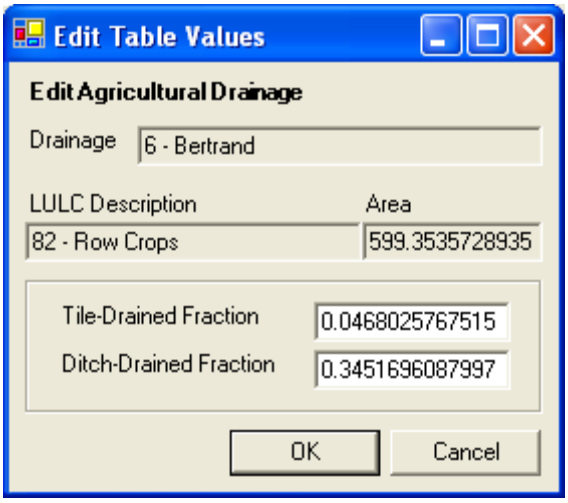

To edit the impervious fraction, change the value in the textbox to the right of Impervious Fraction. To save the impervious fraction that has been entered click the OK button. To discard the impervious fraction that has been entered click the Cancel button.

*Edit Scenario XML File Agricultural Drainage Values*

The Agricultural Drainage form allows the DSS user to edit the Agricultural Drainage values for a WRIA-1 Drainage. These values are then stored in the scenario XML file.



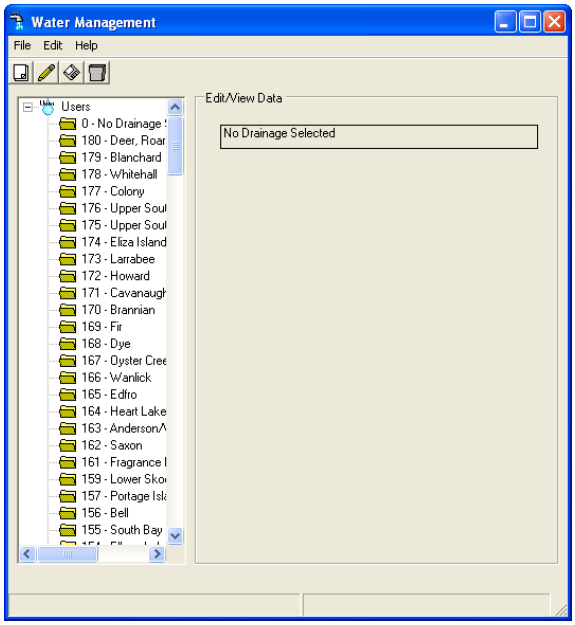
To select a WRIA-1 Drainage click the drop down menu to the right of Drainage. To edit the tile-drained and ditch-drained fractions for a land cover type, double click on the row in the land cover list. The Edit Table Values form will be displayed.



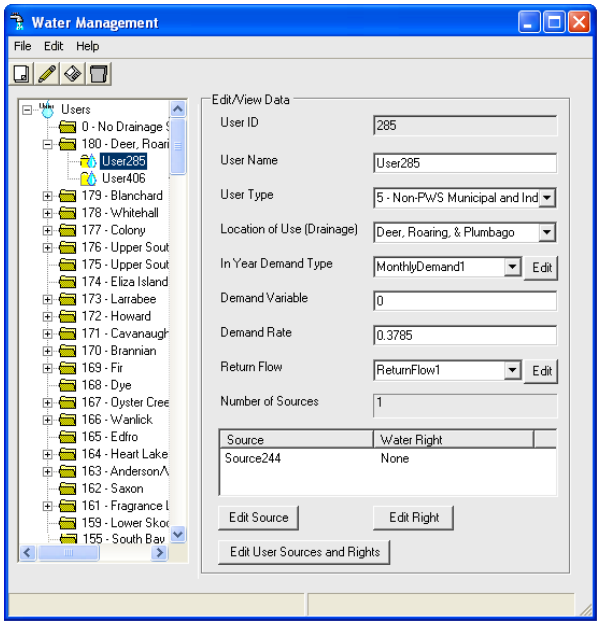

To edit tile-drained fraction, change the text in the textbox to the right of Tile-Drained Fraction. To edit the ditch-drained fraction, change the text in the textbox to the right of Ditch-Drained Fraction. To save values that have been entered click the OK button. To discard the values that have been entered click the Cancel button.

*Edit Water Management*

To edit the Water Management click the ... button to the right of Edit Water Managment in the Water Quality Model Settings form. The Water Management form will be displayed.

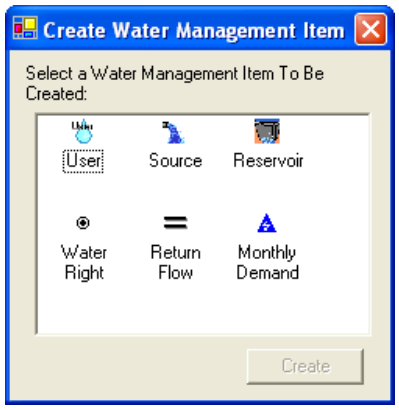



The Water Management form allows the DSS user to manage water users for the WRIA-1 DSS. The water users are grouped by drainage in the list on the left side of the Water Management form. To view the water users located with a WRIA-1 Drainage, click on the icon to expand the drainage folder in the list.



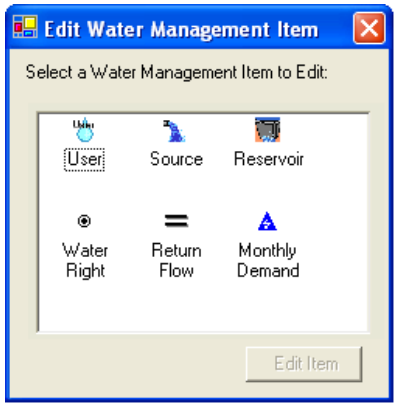

To view a water user's data, click the icon for the desired water user. In the figure above, user 285 has been selected in the list on the left of the Water Management form.

To add a new water management item click the button. The Create Water Management Item form will be displayed.




Select the water management item desired, and then click the Create button. The Edit Data form will be displayed for the water management item created, allowing the DSS user to edit the data values for the new item.

To edit an existing Water Management item click the button. The Edit Water Management Item form will then be displayed.



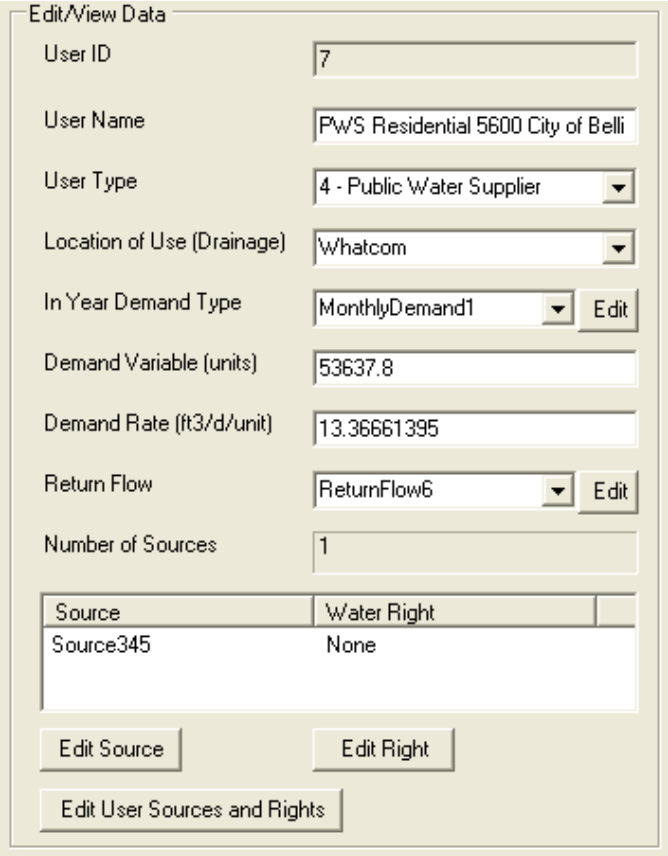

Select the water management item desired, and then click the Edit Item button. The Edit Data form will be displayed for the water management item selected, allowing the DSS user to edit the data values for the item.

To save the Water Management selected click the button. The Water Management items will then be saved to the database.

To delete the water user selected in the water users list click the button. The water user will then be deleted.

*Manage Water Users*

When the DSS user creates a new water user or edits an existing water user, the Edit/View Water User Control is displayed. This allows the DSS user to edit the values for the water user.



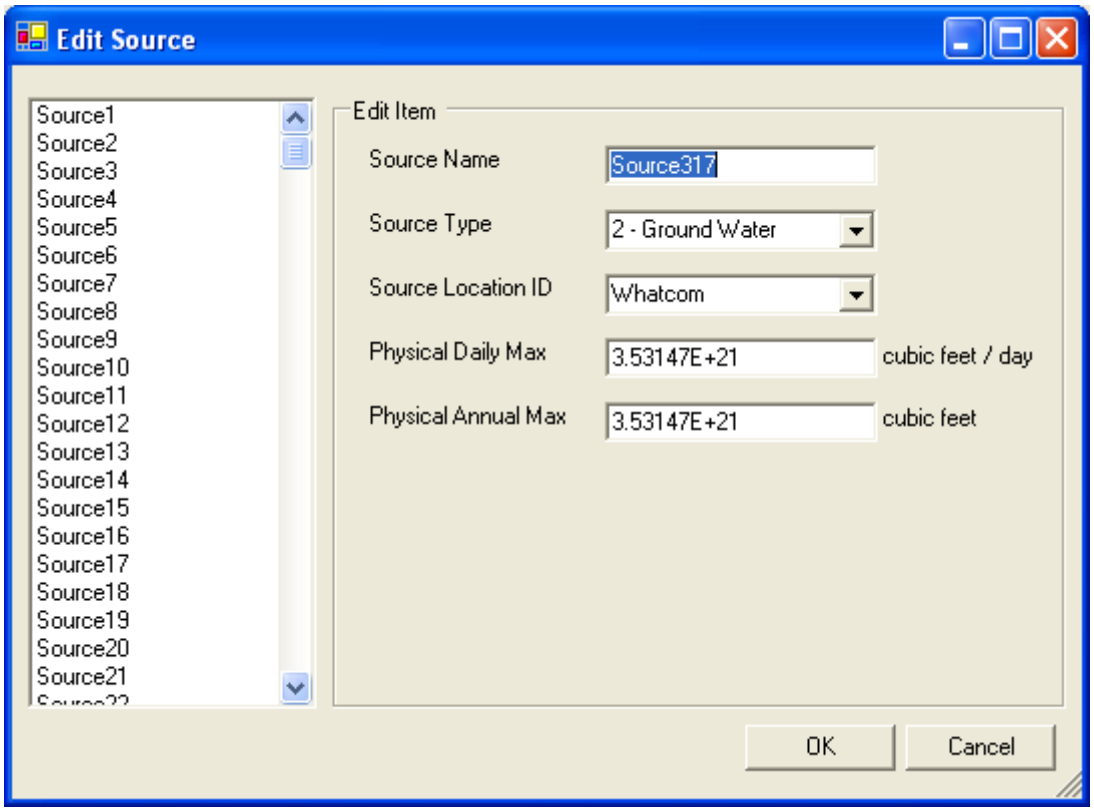

The User ID is a unique identifier of the water user in the Water Management. To edit the user name, edit the text in the text box to the right of User Name. To edit the user type, select a user type from the drop down list to the right of User Type. To edit the location of the user select a location in the drop down list to the right of Location of Use (Drainage). To edit the in year demand type, select a type from the drop down list to the right of In Year Demand Type. To edit the demand variable, edit the text in the text box to the right of Demand Variable. To edit the demand rate, edit the text in the text box to the right of Demand Rate. To edit the return flow select a return flow from the drop down list to the right of Return Flow. The number of sources is listed to the right of Number of Sources.

To edit the source selected in the sources list click the Edit Source button. To edit the right selected in the sources list click the Edit Right button. To edit the user-source-rights information click the Edit User Sources and Rights button.

To use the changed water user values click the OK button. To discard the changed water user values click the Cancel button.

*Manage Water Sources*

When the DSS user creates a new water source or edits an existing water source, the Edit/View Water Source Control is displayed. This allows the DSS user to edit the values for the water source.

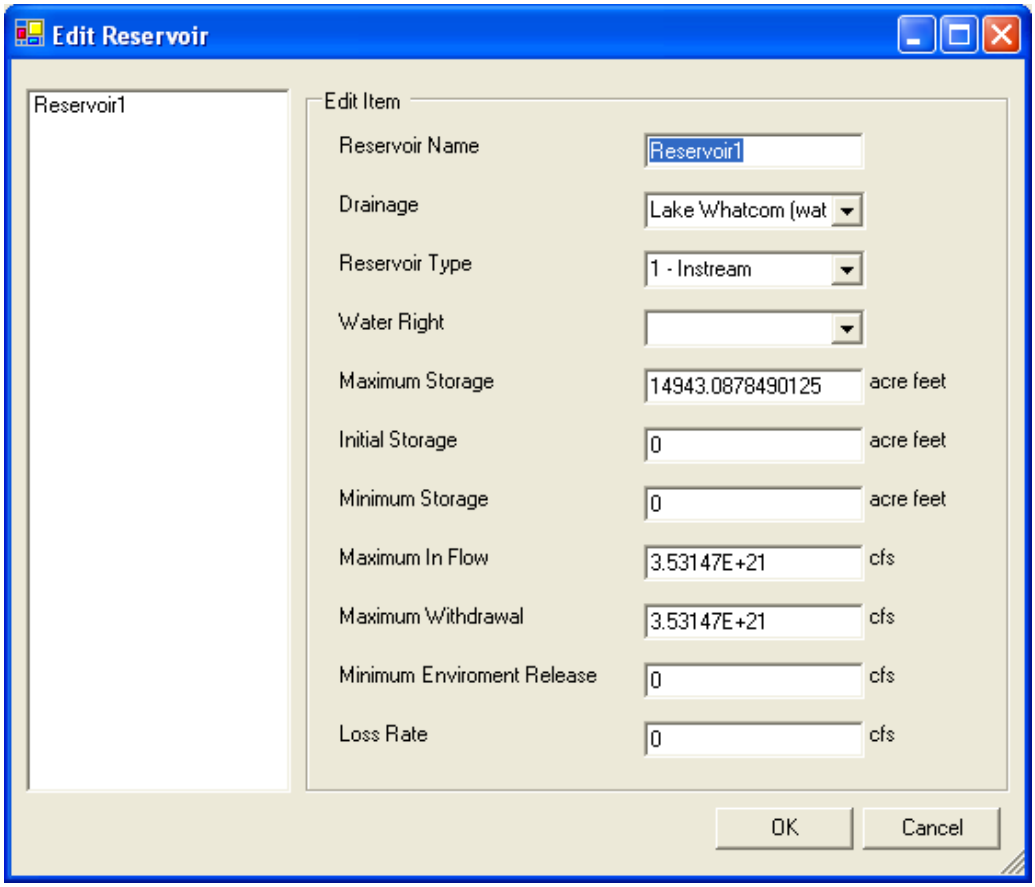



To edit the source name, edit the text in the text box to the right of Source Name. To edit the source type, select a type from the drop down menu to the right of Source Type. To edit the source location, select a location from the drop down menu to the right of Source Location ID. To edit the physical daily maximum withdrawal, edit the text in the text box to the right of Physical Daily Max. To edit the physical annual maximum withdrawal, edit the text in the text box to the right of Physical Annual Max.

To use the changed water source values click the OK button. To discard the changed water source values click the Cancel button.

*Manage Reservoirs*

When the DSS user creates a new reservoir or edits an existing water user, the Edit/View Reservoir Control is displayed. This allows the DSS user to edit the values for the reservoir.

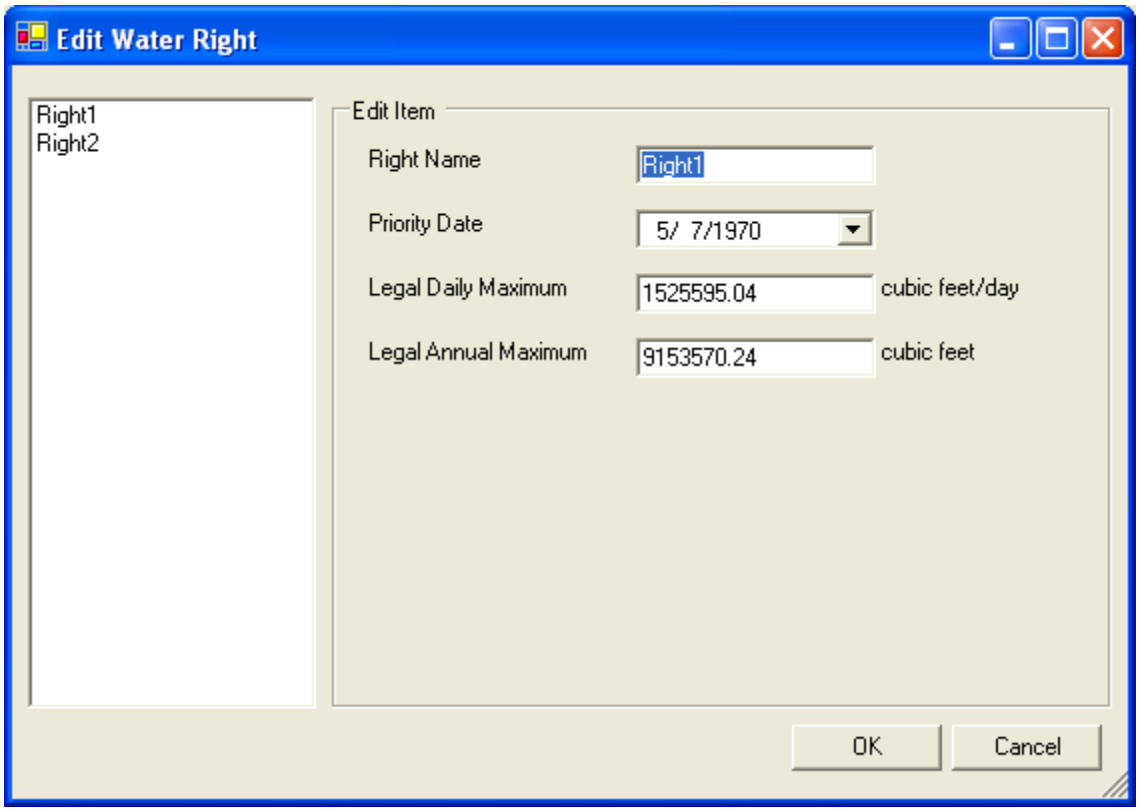



To edit the reservoir name, edit the text in the text box to the right of Reservoir Name. To edit the drainage, select a WRIA-1 Drainage from the drop down list to the right of Drainage. To edit the reservoir type, select a type from the drop down menu to the right of Reservoir Type. To edit the water right, select a right from the drop down menu to the right of Water Right. To edit the maximum storage, edit the text in the text box to the right of Maximum Storage. To edit the initial storage, edit the text in the text box to the right of Initial Storage. To edit the minimum storage, edit the text in the text box to the right of Minimum Storage. To edit the maximum in flow, edit the text in the text box to the right of Maximum In Flow. To edit the maximum withdrawal, edit the text in the text box to the right of Maximum Withdrawal. To edit the minimum environment release, edit the text in the text box to the right of Minimum Enviroment Release. To edit the loss rate, edit the text in the text box to the right of Loss Rate.

To use the changed reservoir values click the OK button. To discard the changed reservoir values click the Cancel button.

*Manage Water Rights*

When the DSS user creates a new water right or edits an existing water right, the Edit/View Water Right Control is displayed. This allows the DSS user to edit the values for the water right.

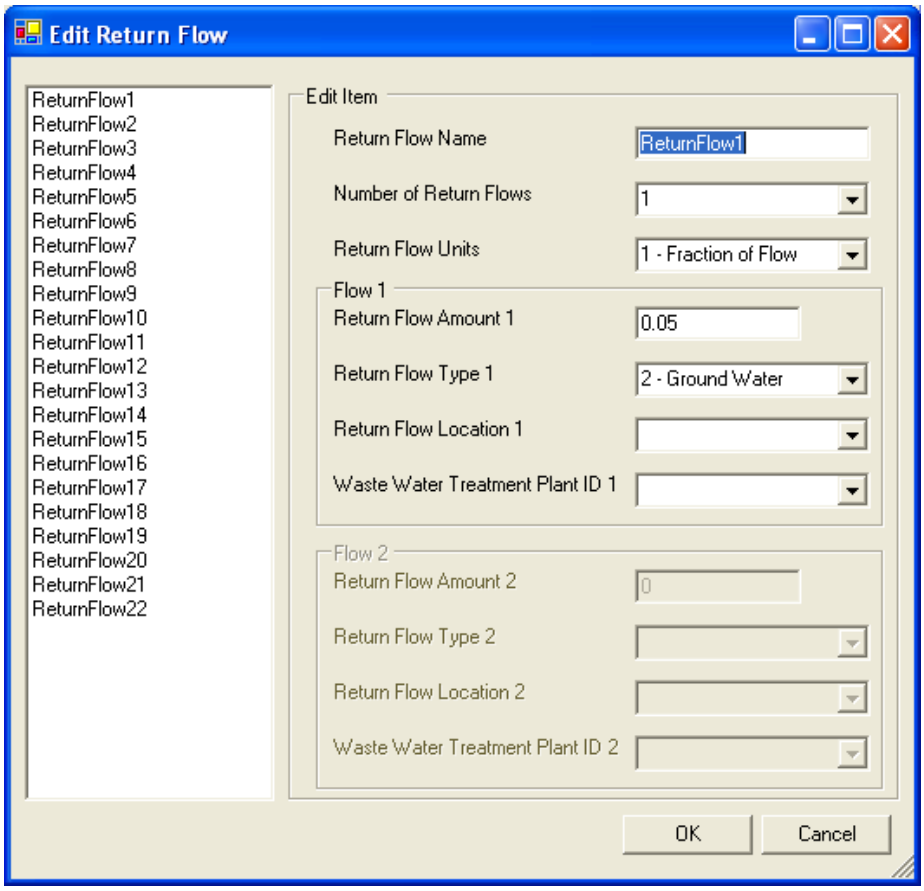



To edit the water right name, edit the text in the text box to the right of Right Name. To edit the priority date, click the drop down menu to the right of Priority Date. To edit the legal daily maximum allowance, edit the text in the text box to the right of Legal Daily Maximum. To edit the legal annual maximum allowance, edit the text in the text box to the right of Legal Annual Maximum.

To use the changed water right values click the OK button. To discard the changed water right values click the Cancel button.

*Manage Return Flows*

When the DSS user creates a new return flow or edits an existing water user, the Edit/View Return Flow Control is displayed. This allows the DSS user to edit the values for the return flow.



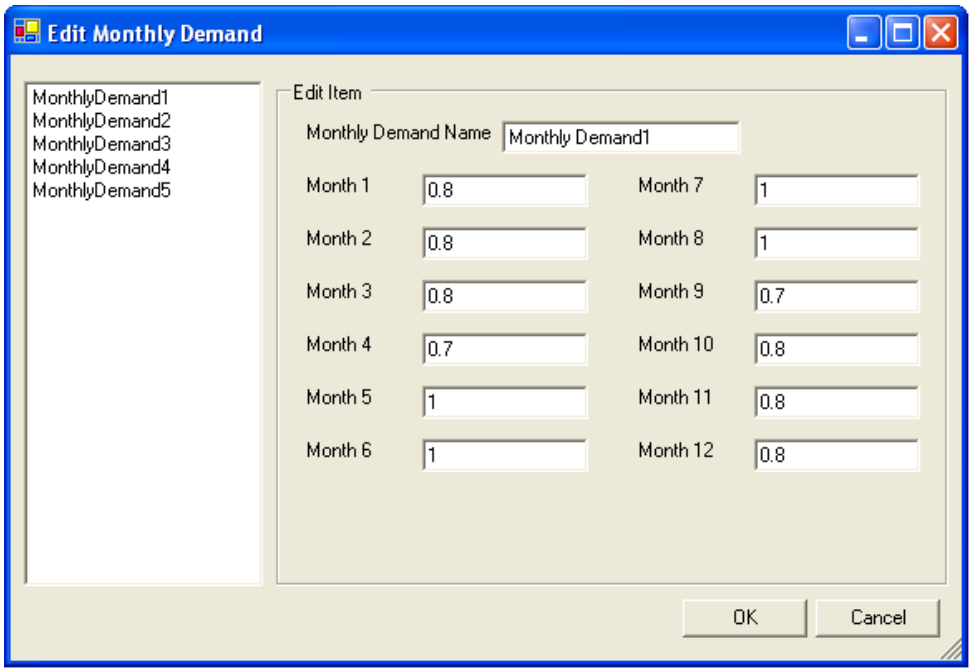

To edit the return flow name, edit the text in the text box to the right of Return Flow Name. To edit the number of returns select a number from the drop down menu to the right of Number of Return Flows. To edit the return flow units, select a unit from the drop down menu to the right of Return Flow Units.

There are two possible locations associated with a return flow. Each has a separate set of inputs. To edit the amount of flow, edit the text in the text box to the right of Return Flow Amount 1. Note that amount refers to the return flow units above, which may be fractional, so amount does not in general have units. To edit the flow type for the associated return amount, select a type from the drop down menu to the right of Return Flow Type 1. To edit the flow location for the associated return amount, select a location from the drop down menu to the right of Return Flow Location 1. To edit the wastewater treatment plant associated with the flow amount, select a WWTP from the drop down list to the right of Waste Water Treatment Plant ID 1.

To use the changed return flow values click the OK button. To discard the changed return flow values click the Cancel button.

*Manage Monthly Demand*

When the DSS user creates a new monthly demand or edits an existing monthly demand, the Edit/View Monthly Demand Control is displayed. This allows the DSS user to edit the values for the monthly demand.

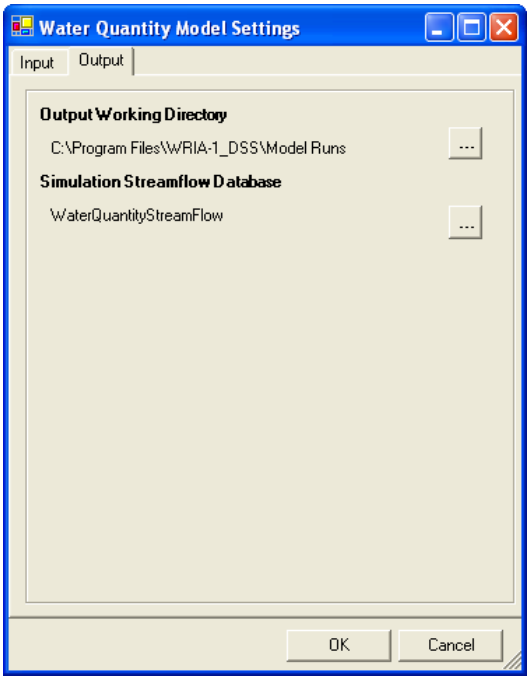



To edit the monthly demand name, edit the text in the text box to the right of Monthly Demand Name. To edit the monthly demand values for a month, edit the text in the text box to the right of Month 1, for Month 1 in this example.

To use the changed monthly demand values click the OK button. To discard the changed monthly demand values click the Cancel button.

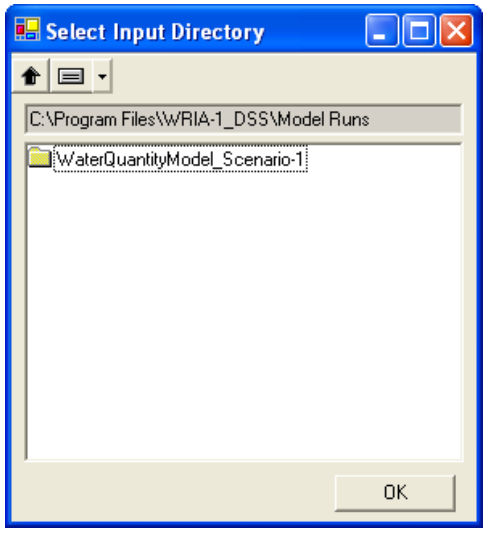
***Edit Water Quantity Model Outputs***

The outputs tab of the Water Quantity Model Settings allows the DSS user to edit the outputs for the Water Quantity Model.




*Select Water Quantity Output Directory*

To select an output directory for the Water Quantity Model click the ... button to the right of **Output Working Directory**. The Select Directory form will then be displayed. The Select Input Directory form allows the DSS user to select a directory that will contain the output folder for the Water Quantity Model. The default location for the TopNet Input Directory is "C:\Program Files\WRIA-1_DSS\ModelRuns". When the Water Quantity Model is run in the DSS Scenario Builder, a directory will be created in the directory selected. If the scenario id were 99, the output directory path could be "C:\Program Files\WRIA-1_DSS\ModelRuns\WaterQuantityModel_Scenario99".

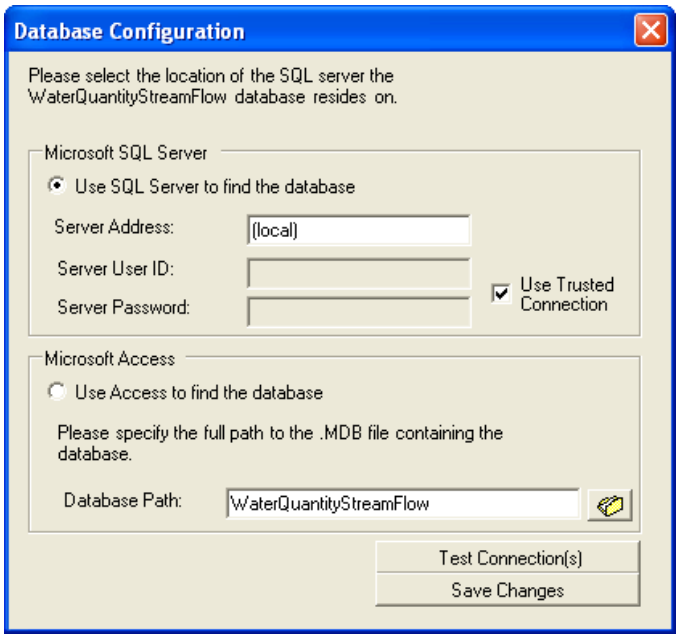


To move up a directory level click the button. To change the drive to browse for folders click the button. To select a folder as the desired directory, click on the folder's name. To open a folder double-click the folder's name.

To save the selected directory, click the OK button.

*Select Simulation Stream Flow Database*

To select the Simulated Stream Flow Database, click the ... button to the right of Simulation Streamflow Database.



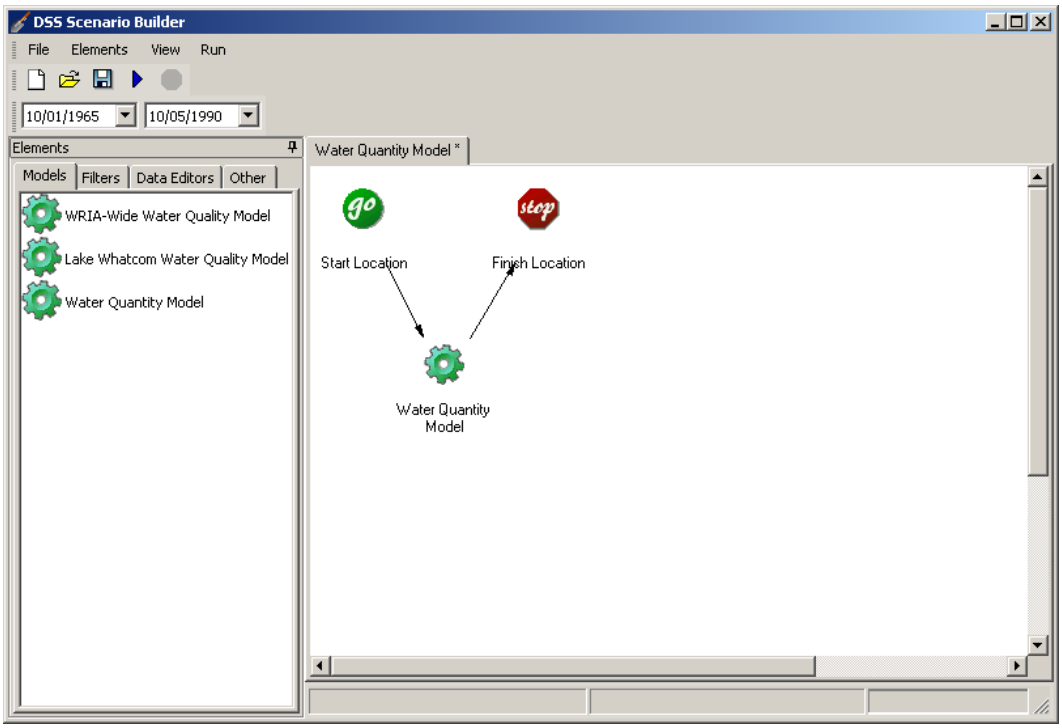

To change the address of the Microsoft SQL Server containing the WaterQuantityStreamFlow Database, enter the server address in the textbox to the right of Server Address:. To enter the User ID for the server, uncheck Use Trusted Connection and enter the user id in the textbox to the right of Server User ID:. To enter the Server Password, uncheck Use Trusted Connection and enter the password in the textbox to the right of Server Password:.

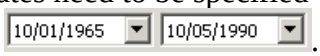
The default location for the server installed with the WRIA-1 DSS is "(local)", with the Use Trusted Connection check box checked.

To save the selected database location, click the Save Changes button.

## Run Water Quantity Model

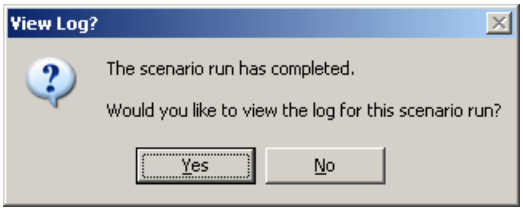
The DSS Scenario Builder allows the DSS user to run the Water Quantity Model.

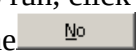



Before the Water Quantity Model can be run, the start and end dates need to be specified in the Scenario Builder. To change the start and end dates, click  . The start date is the date listed to the left, and the end date is the date listed on the right.

To run the scenario, click the button. If you would like to abort the scenario run, click the button to cancel the scenario run.

When the scenario has completed, the DSS user will be asked if they would like to view the scenario run log (a text file containing messages output by the elements contained in the scenario that has just been run).



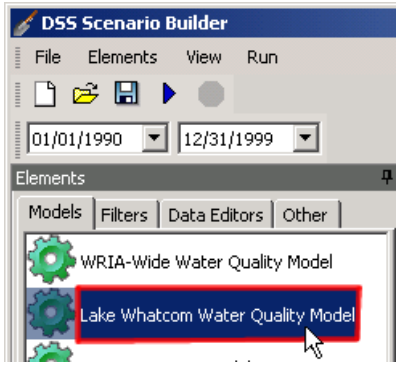

To view the log for the scenario run, click the button. To decline viewing the scenario log at this time click the  button.

# Lake Whatcom Water Quality Model Interface

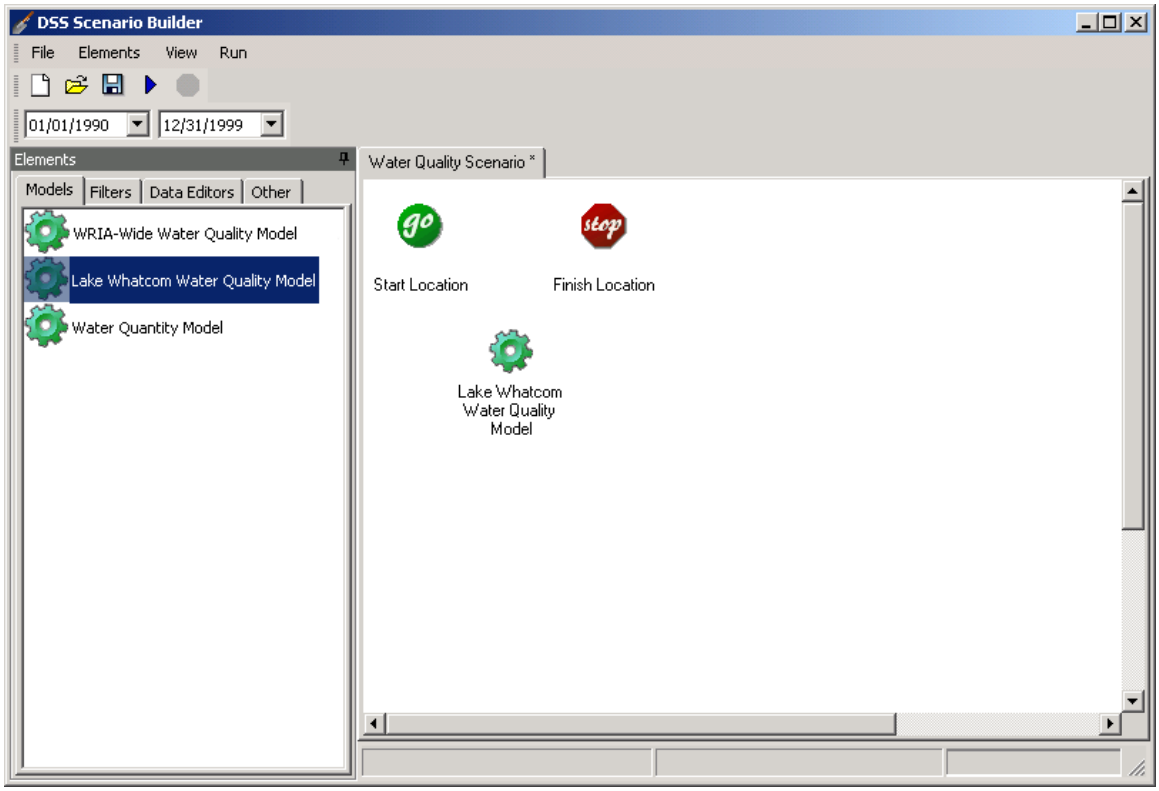
The Lake Whatcom Water Quality Model Interface allows the DSS user to specify the input for the Water Quality Model for the Lake Whatcom drainage.

## Add Lake Whatcom Water Quality Model to DSS Scenario

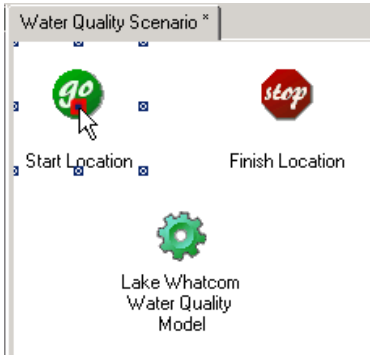
To add the Lake Whatcom Water Quality Model to a DSS scenario layout, first open the DSS Scenario Builder (See Model Manger's User Guide). Create a new scenario layout by clicking . Click on the Models tab in the Scenario Builder and then double click on the Lake Whatcom Water Quality Model highlighted in red below.



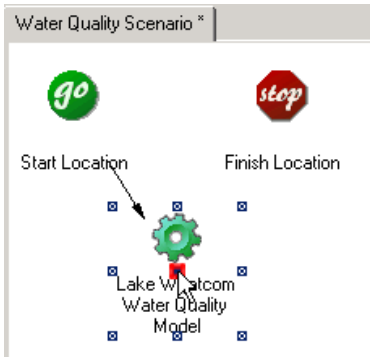

The Water Quality Model will then be added to the scenario layout.




To prepare the Lake Whatcom Water Quality Model to be run in the Scenario Builder, you must connect the model to the Start Location and the Finish Location nodes in the scenario layout.

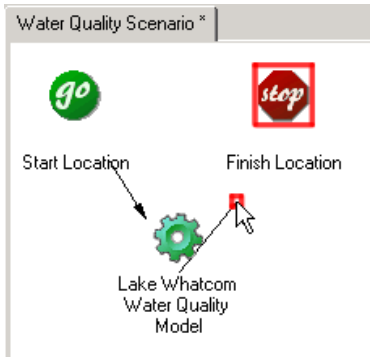



First, click on the Start Location node. Then, click and hold the mouse on the blue box in the center of the node's icon. While still pressing the mouse button, drag the mouse to the Lake Whatcom Water Quality Model node in the scenario layout.



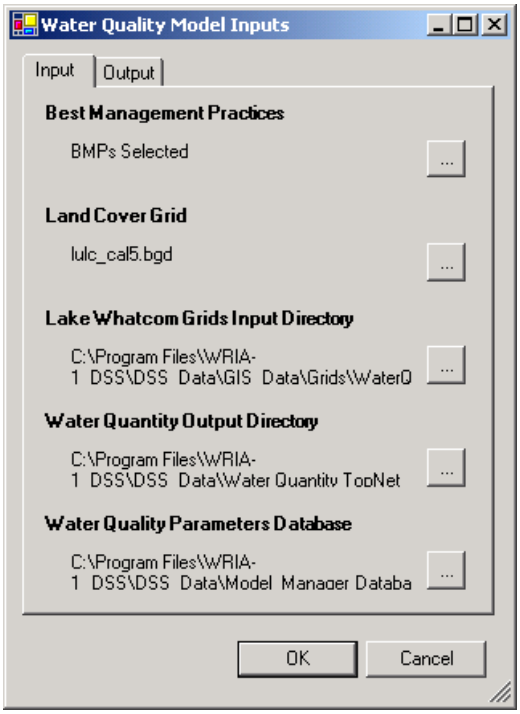

Repeat the previous step by again clicking the blue box in the center of the Lake Whatcom Water Quality Model, dragging the mouse to the node.






The Lake Whatcom Water Quality Model is now ready to be run in the DSS Scenario Builder. To edit the Lake Whatcom Water Quality Model settings, double click on the Lake Whatcom Water Quality Model node in the scenario layout.

The Lake Whatcom Water Quality Model settings form will be displayed as seen below.

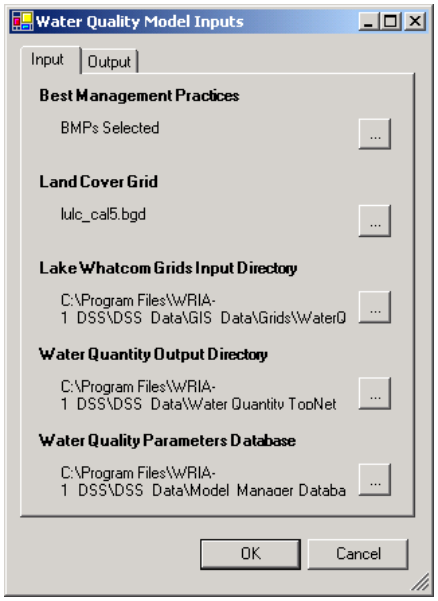



## Edit Lake Whatcom Water Quality Model Settings

The Lake Whatcom Water Quality Inputs form allows the user to edit/view the settings for the model. To edit/view the input settings, click the Input tab. To edit/view the output settings, click the Output tab.

### *Edit Lake Whatcom Water Quality Model Inputs*

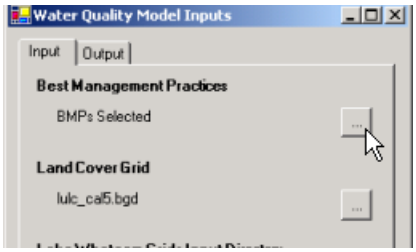
The input tab allows the user to view/edit the input settings for the Lake Whatcom Water Quality Model. To change the settings for a particular input, select the ... button to the right of the input setting.



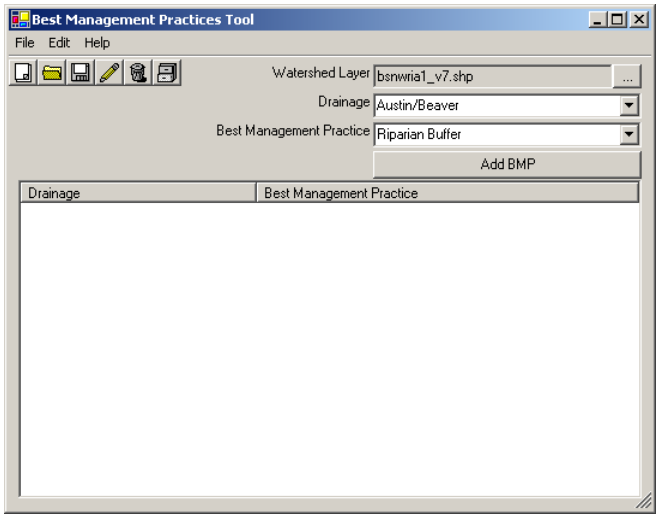

To save the changes to the settings that have been made in the input form, click the OK button. To cancel the changes to the settings click the Cancel button.

*Select Best Management Practices*

The Best Management Practices Tool allows the DSS user to select Best Management Practices (BMP) for multiple drainages to be used by the Water Quality Model as an input. To access this tool press the ... button to the right of **Best Management Practices** on the Input tab of the Water Quality Model Inputs window as shown below.

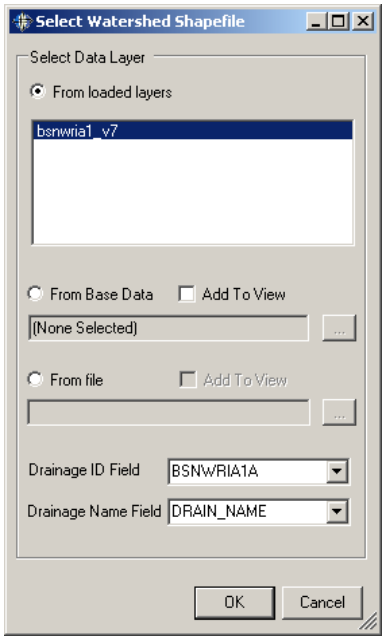



This will bring up the Best Management Practices Tool as shown below.




*Select Watershed Layer*

To select a Watershed Layer click the ... button to next to the Watershed Layer box. The Select Layer form will then be displayed.




To select a shapefile loaded in MapWindow's view click the From loaded layers radio button, then select the name of the shapefile layer in the layer list.

To select a shapefile from the DSS base data click the From Base Data radio button, then click the ... button. A base data form will be displayed with Drainages shown. Click the OK button.

To select a shapefile from file click the From file radio button, then click the ... button. An open file dialog will be displayed. Browse to a polygon shapefile and click open.

To select the shapefile field to be used for the id, choose a field listed in the dropdown list to the right of Drainage ID Field. To select the shapefile field to be used for the name, shoose a field listed in the dropdown list to the right of Drainage Name Field.

To save the polygon shapefile that has been selected click the OK button. To discard the polygon shapefile that has been selected click the Cancel button.

*Select Drainage*

To select a drainage contained in the Watershed Layer, choose a drainage in the drop down list to the right of Drainage on the Best Management Practice Tool.

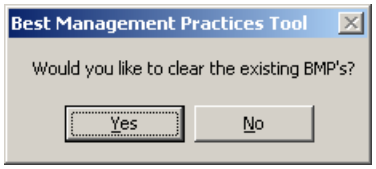
*Select Best Management Practice*

To select a Best Management Practice, choose a Best Management Practice from the drop down list to the right of Best Management Practice on the Best Management Practice Tool.

To add the Best Management Practice specified in the drop down list, click the Add BMP button. The Best Management Practice for the selected drainage will be added to the Best Management Practice list.

*Create New Best Management Practices*

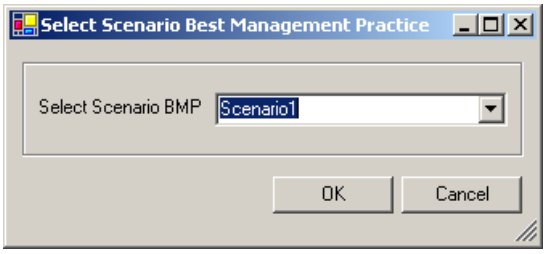
To create a new set of BMP's, click the button. You will be asked to confirm whether you would like to start with new BMP's, discarding any that may currently be displayed in the BMP list.




Click the Yes button to clear existing BMP's. Click the No button to keep the current list of BMP's.

*Opening Existing Best Management Practices*

To open a set of previously defined BMP click the button. The Select Scenario form will be displayed, allowing the DSS user to select a set of BMP's associated with a DSS Scenario Run ID.



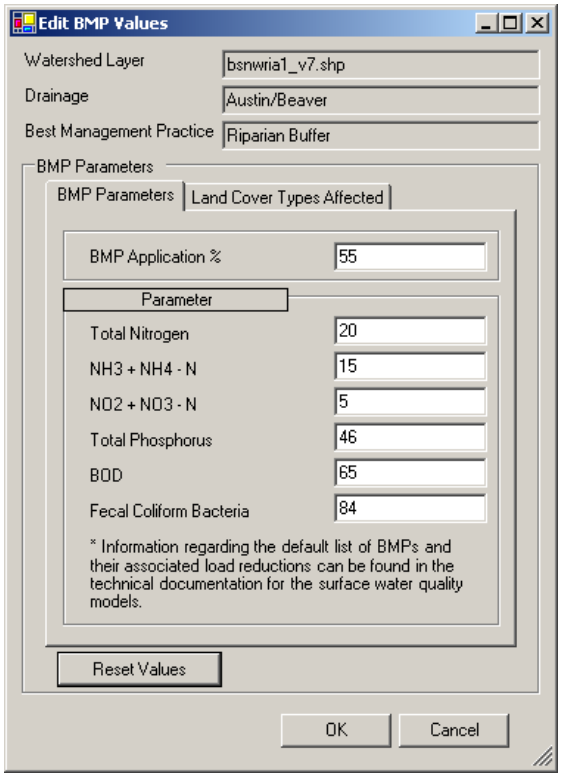

To open the BMP's contained in the selected scenario click the OK button. To cancel opening a set of BMP's from a previous scenario click the Cancel button.

*Save Best Management Practices*

To save the BMP's displayed in the BMP list click the button. This saves the BMP's displayed in the BMP list for the Water Quality Model in the current scenario layout.

*Edit Best Management Practices*

To edit the BMP selected in the BMP list click the button. The Edit BMP Values form will then be shown for the BMP selected in the BMP list.




To change a value for the BMP click on the BMP Parameters tab. To change a particular value for the BMP, enter a number between 0 and 100 into the text box to the right of the value you wish to edit. To change whether a land cover type is affected by the BMP, click on the Land Cover Types Affected tab. To change whether a particular land cover type is affected by the BMP check or uncheck the ☑ box to the left of the land cover type.

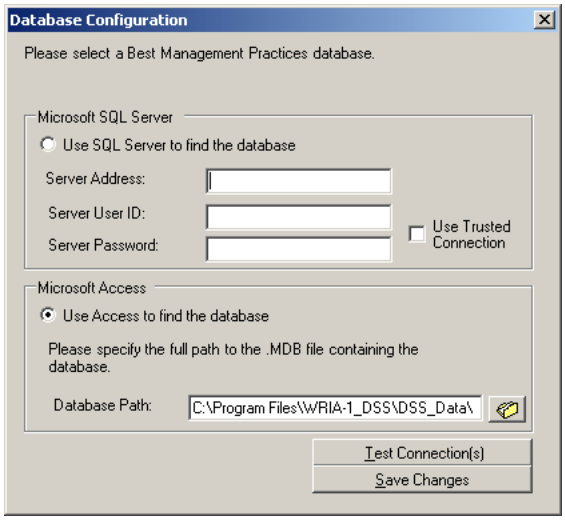
To use the changed BMP values click the OK button. To discard the changed BMP values click the Cancel button.

*Delete Best Management Practices*

To delete the BMP selected in the BMP list click the button. This removes the selected BMP from the BMP list.

*Change Best Management Practices Database*

To change the BMP database, click the button. The Database Configuration form will be displayed, allowing the user to browse to the Best Management Practices Database.

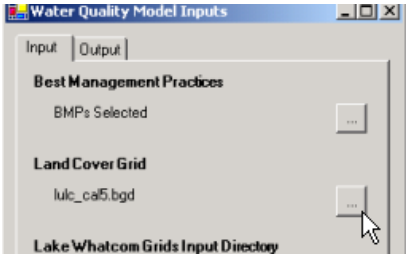



To browse to a Microsoft Access Best Management Practices Database click the button. An open file dialog will be displayed. The default location for the Best Management Practices is "C:\Program Files\WRIA-1_DSS\DSS_Data\Model Manager Databases\BestManagementPractices.mdb".

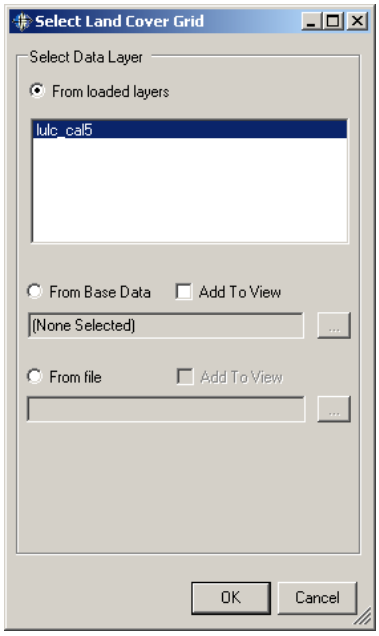
To save the selected database location, click the Save Changes button.

*Select Land Cover Grid*

The Select Land Cover Grid form allows a DSS user to select a land cover grid for the Water Quality Model. To access this form press the ... button to the right of **Land Cover Grid** on the Input tab of the Water Quality Model Inputs window as shown below.




This will open the Select Land Cover Grid from as shown below. The default location for the WRIA-1 Land Cover Grid is "C:\Program Files\WRIA-1_DSS\DSS_Data\GIS_Data\Grids\LandUse\lulc_cal5.bgd".




To select a grid loaded in MapWindow's view click the From loaded layers radio button, and then select the name of the grid layer in the layer list.

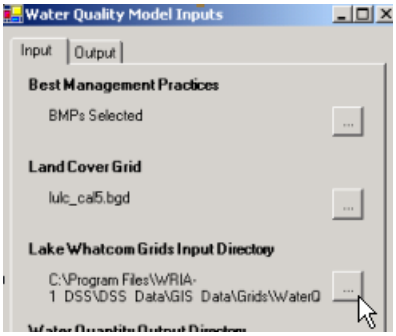
To select a grid from the DSS base data click the From Base Data radio button, then click the ... button. A base data form will be displayed with "LandCover" shown. Click the OK button.

To select a grid from file click the From file radio button, then click the ... button. An open file dialog will be displayed. Browse to a land cover grid and click open.

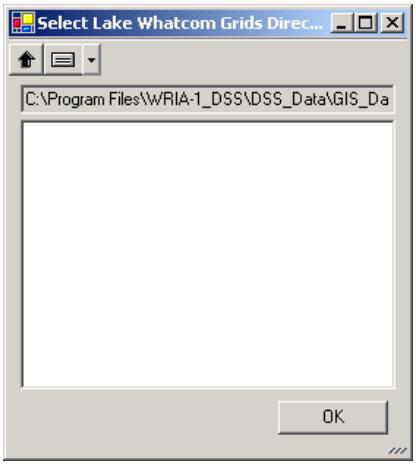
To save the land cover grid that has been selected click the OK button. To discard the land cover grid that has been selected click the Cancel button.

*Select Lake Whatcom Grids Directory*

The Select Directory form allows the DSS user to select a directory containing the Lake Whatcom grids. To access this form press the ... button to the right of **Lake Whatcom Grids Inputs Directory** on the Inputs tab of the Water Quality Model Inputs window as shown below.




This will bring up the Select Lake Whatcom Grids Inputs Directory form as shown below. The default location for the Lake Whatcom Grids is "C:\Program Files\WRIA-1_DSS\DSS_Data\GIS_Data\Grids\WaterQualityGrids".

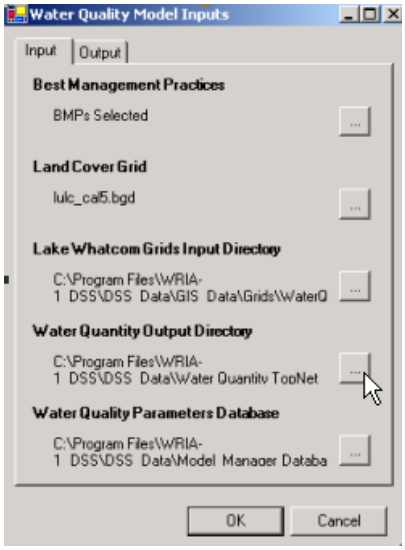



To move up a directory level click the button. To change the drive to browse for folders click the button. To select a folder as the desired directory, click on the folder's name. To open a folder double-click the folder's name.

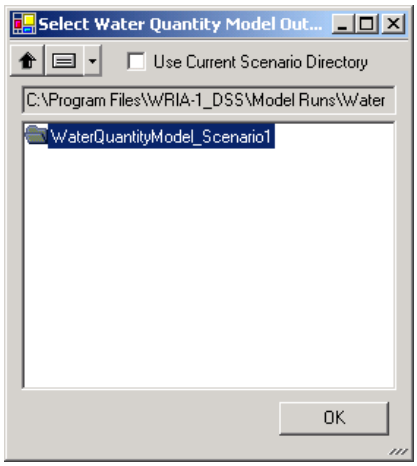
To save the selected directory, click the OK button.

*Select Water Quantity Output Directory*

The Select Directory form allows the DSS user to select a directory containing the Water Quantity Model output. To access this form press the button to the right of **Water Quantity Output Directory** on the Input tab of the Water Quality Model Inputs window as shown below.




This will open the Select Water Quality Model Output Directory form as shown below. The default location for the Water Quantity Model output is "C:\Program Files\WRIA-1_DSS\Water Quantity TopNet Files\Existing".



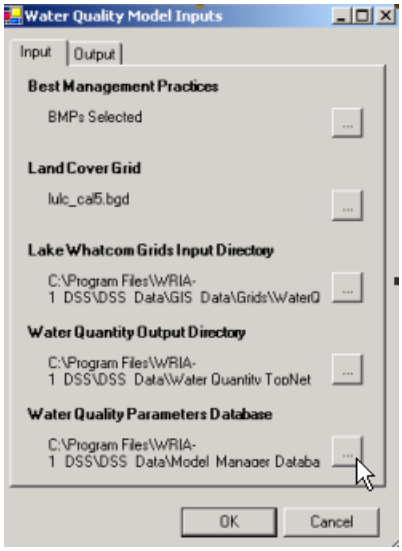

To move up a directory level click the button. To change the drive to browse for folders click the button. To use a directory that will be created when the current scenario is run, check the Use Current Scenario Directory box. To select a folder as the desired directory, click on the folder's name. To open a folder double-click the folder's name.

To save the selected directory, click the OK button.

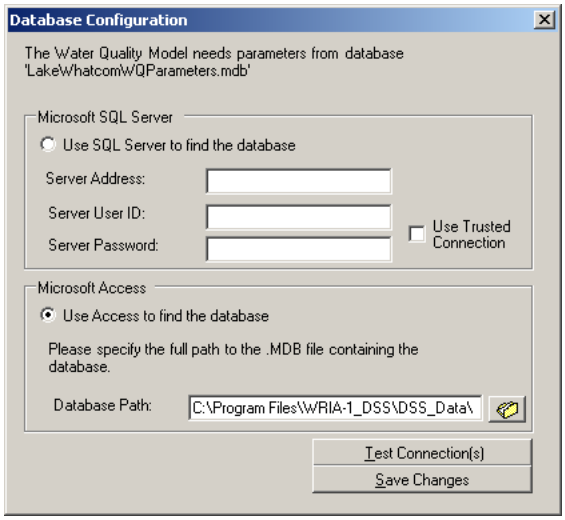
*Select Water Quality Parameters Database*

The Database Configuration form allows a DSS user to select a Water Quality Parameters Database. To access this form press the button to the right of **Water Quality**

**Parameters Database** on the Input tab of the Water Quality Model Inputs window as shown below.




This will bring up the Database Configuration form as shown below.

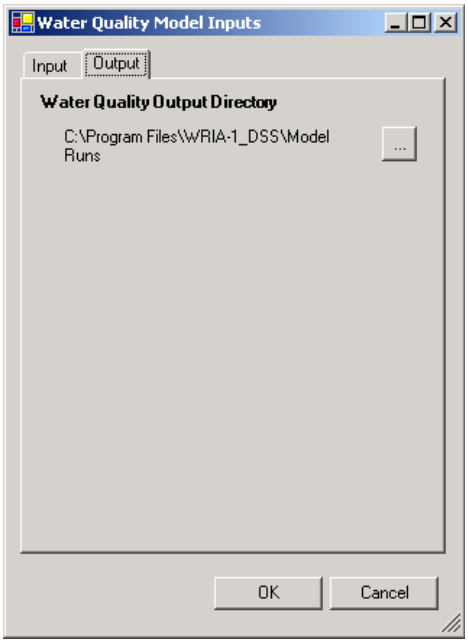



To browse to a Microsoft Access Water Quality Parameters database click the button. An open file dialog will be displayed. The default location for the Lake Whatcom Water Quality Parameters Database is "C:\Program Files\WRIA-1_DSS\DSS_Data\Model Manager Databases\LakeWhatcomWQParameters.mdb".

To save the selected database location, click the button.

### *Edit Lake Whatcom Water Quality Model Outputs*

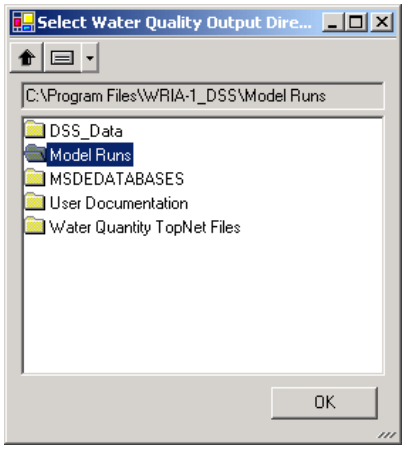
The output tab allows the user to view/edit the output settings for the Lake Whatcom Water Quality Model.




To save the changes to the settings that have been made in the outputs form, click the OK button. To cancel the changes to the settings click the Cancel button.

*Select Water Quality Output Directory*

The Select Directory form allows the DSS user to select a directory for the Water Quality Model output. To access this form, select the Output tab in the Water Quality Model Inputs form. Press the ... button to the right of **Water Quality Output Directory** on the Output tab of the Water Quality Model Inputs form. The default location for the Water Quality Model output is "C:\Program Files\WRIA-1_DSS\Model Runs".

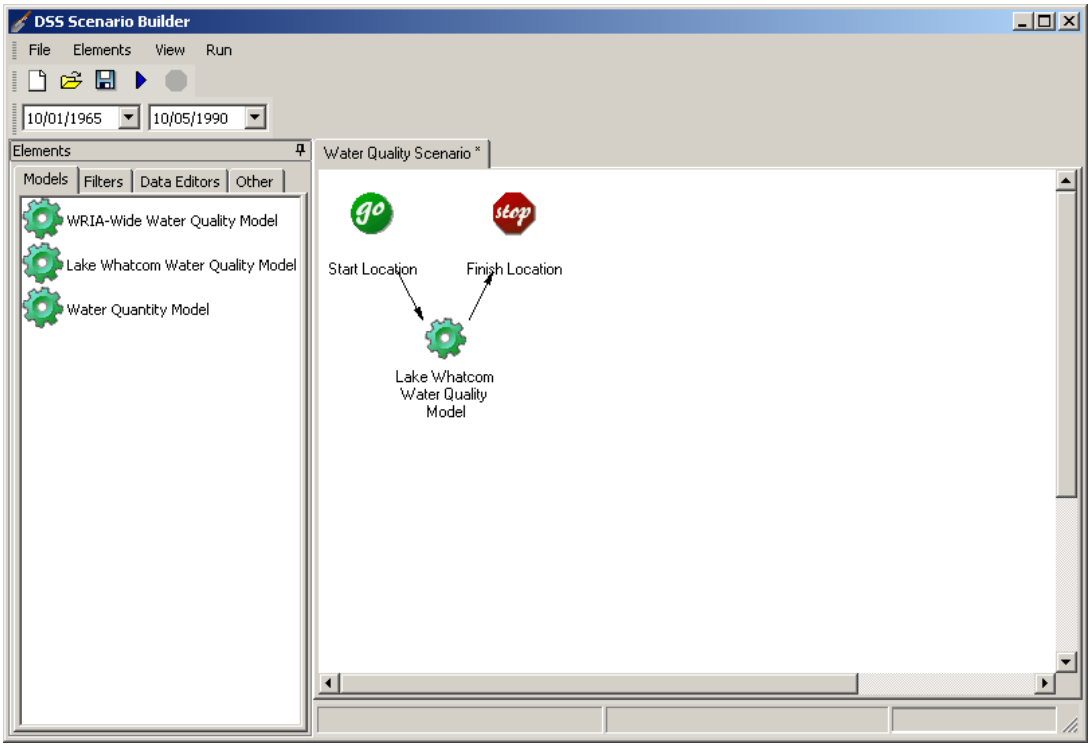


To move up a directory level click the button. To change the drive to browse for folders click the button. To select a folder as the desired directory, click on the folder's name. To open a folder double-click the folder's name.

To save the selected directory, click the OK button.

## Run Lake Whatcom Water Quality Model

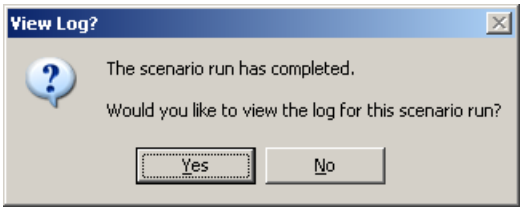
The DSS Scenario Builder allows the DSS user to run the Water Quality Model.




Before the Water Quality Model can be run, the start and end dates need to be specified in the Scenario Builder. To change the start and end dates, click 10/01/1965 10/05/1990. The start date is the date listed to the left, and the end date is the date listed on the right.

To run the scenario, click the button. If you would like to abort the scenario run, click the button to cancel the scenario run.

When the scenario has completed, the DSS user will be asked if they would like to view the scenario run log (a text file containing messages output by the elements contained in the scenario that has just been run).



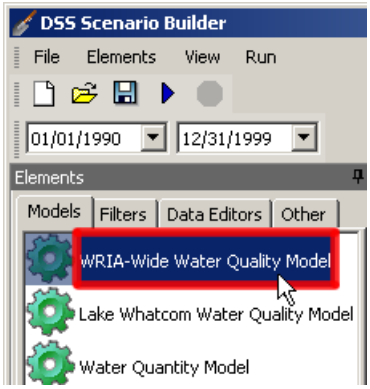

To view the log for the scenario run, click the Yes button. To decline viewing the scenario log at this time click the No button.

# WRIA-Wide Water Quality Model Interface

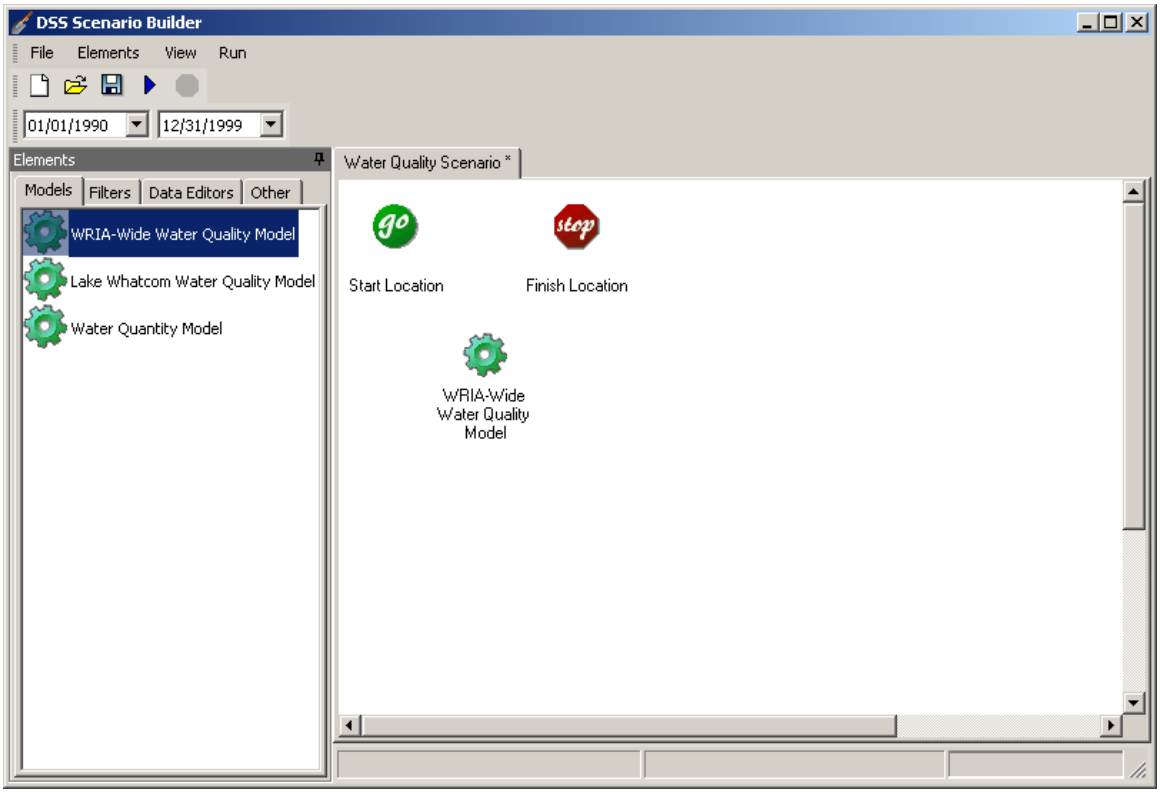
The WRIA-Wide Water Quality Model Interface allows the DSS user to specify the input for the Water Quality Model for all of the WRIA-1 Drainages.

## Add WRIA-Wide Water Quality Model to DSS Scenario

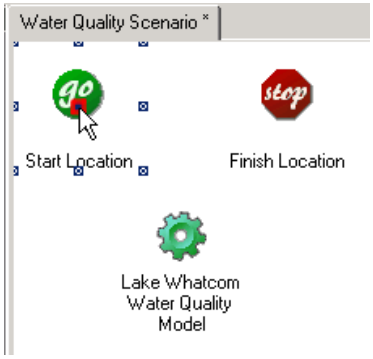
To add the WRIA-Wide Water Quality Model to a DSS scenario layout, first open the DSS Scenario Builder (See Model Manger's User Guide). Create a new scenario layout by clicking . Click on the Models tab in the Scenario Builder and then double click on the WRIA-Wide Water Quality Model highlighted in red below.



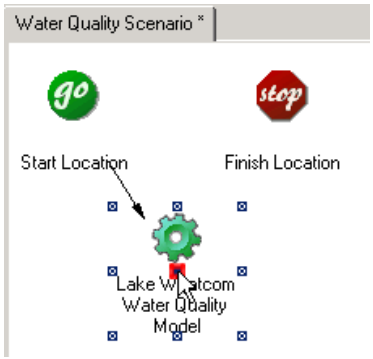

The Water Quality Model will then be added to the scenario layout.



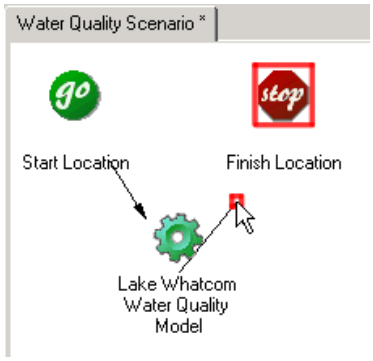

To prepare the WRIA-Wide Water Quality Model to be run in the Scenario Builder, you must connect the model to the Start Location and the Finish Location nodes in the scenario layout.




First, click on the Start Location node. Then, click and hold the mouse on the blue box in the center of the node's icon. While still pressing the mouse button, drag the mouse to the WRIA-Wide Water Quality Model node in the scenario layout.

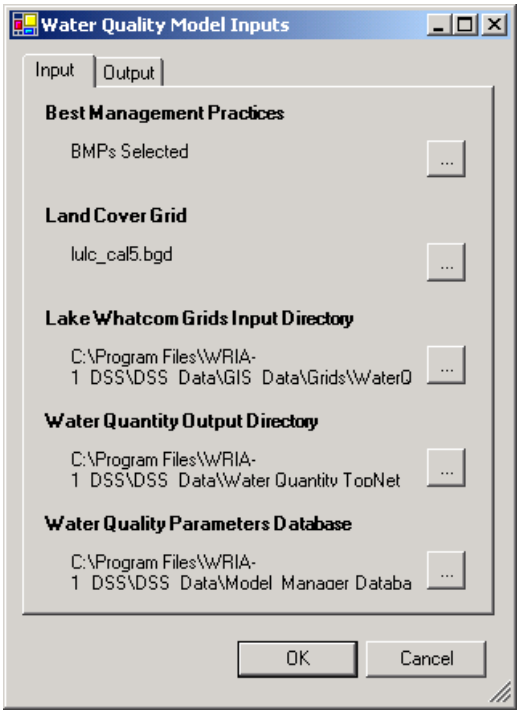



Repeat the previous step by again clicking the blue box in the center of the WRIA-Wide Water Quality Model, dragging the mouse to the Finish Location node.




The WRIA-Wide Water Quality Model is now ready to be run in the DSS Scenario Builder. To edit the WRIA-Wide Water Quality Model settings, double click on the WRIA-Wide Water Quality Model node in the scenario layout.

The WRIA-Wide Water Quality Model settings form will be displayed as seen below.

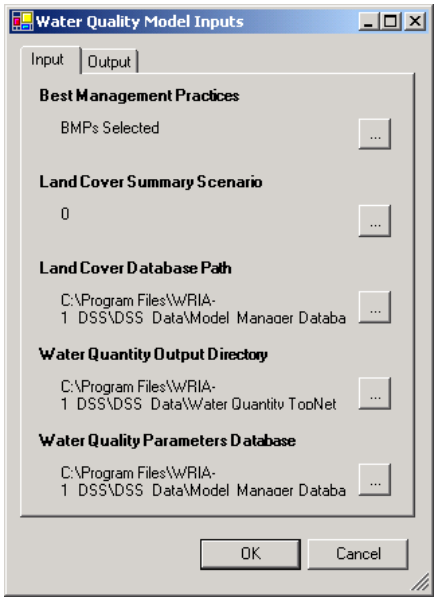



## Edit WRIA-Wide Water Quality Model Settings

The WRIA-Wide Water Quality Inputs form allows the user to edit/view the settings for the model. To edit/view the input settings, click the Input tab. To edit/view the output settings, click the Output tab.

### *Edit WRIA-Wide Water Quality Model Inputs*

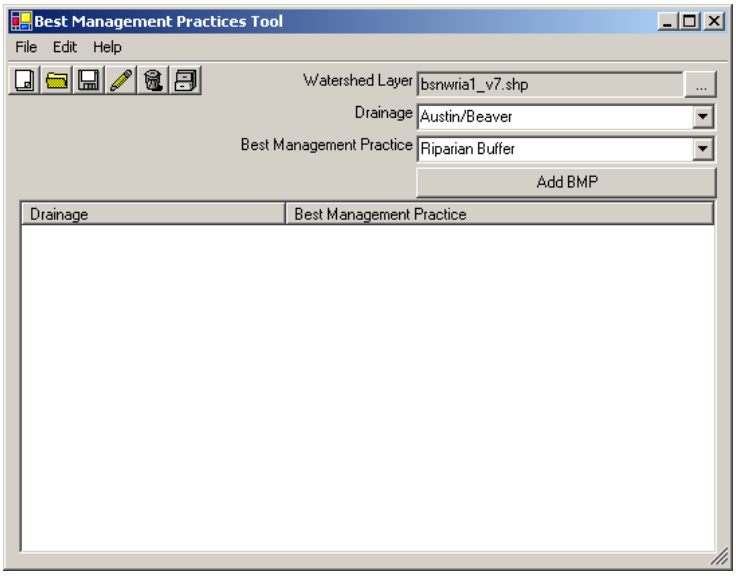
The input tab allows the user to view/edit the input settings for the WRIA-Wide Water Quality Model. To change the settings for a particular input, select the ... button to the right of the input setting.




To save the changes to the settings that have been made in the inputs form, click the OK button. To cancel the changes to the settings click the Cancel button.

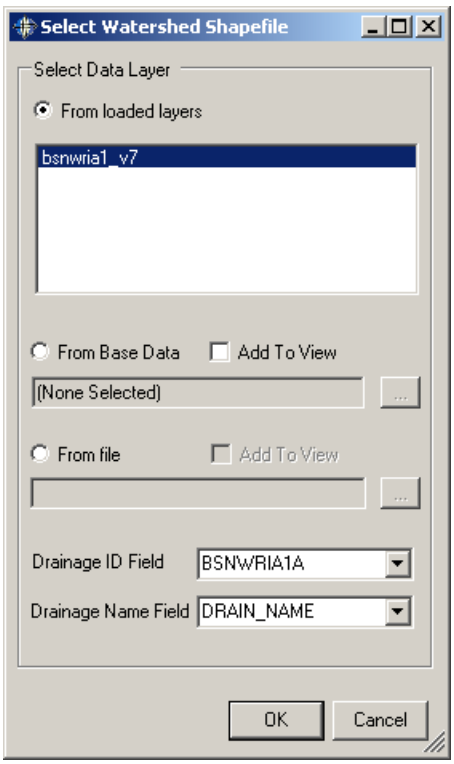
*Select Best Management Practices*

The Best Management Practices Tool allows the DSS user to select Best Management Practices (BMP) for multiple drainages to be used by the Water Quality Model as an input. To access this tool, select the ... button to the right of **Best Management Practices** on the Input tab of the Water Quality Model Inputs form. This will bring up the Best Management Practices Tool form as shown below.



*Select Watershed Layer*

To select a Watershed Layer click the ... button to next to the Watershed Layer box in the Best Management Practices Tool form. The Select Layer form will then be displayed as shown below.




To select a shapefile loaded in MapWindow's view click the From loaded layers radio button, then select the name of the shapefile layer in the layer list.

To select a shapefile from the DSS base data click the From Base Data radio button, then click the ... button. A base data form will be displayed with Drainages shown. Click the OK button.

To select a shapefile from file click the From file radio button, then click the ... button. An open file dialog will be displayed. Browse to a polygon shapefile and click open.

To select the shapefile field to be used for the id, choose a field listed in the dropdown list to the right of Drainage ID Field. To select the shapefile field to be used for the name, shoose a field listed in the dropdown list to the right of Drainage Name Field.

To save the polygon shapefile that has been selected click the OK button. To discard the polygon shapefile that has been selected click the Cancel button.

*Select Drainage*

To select a drainage contained in the Watershed Layer, choose a drainage in the drop down list to the right of Drainage in the Best Management Practices Tool form.

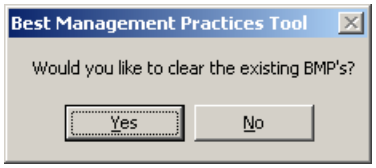
*Select Best Management Practice*

To select a Best Management Practice, choose a Best Management Practice from the drop down list to the right of Best Management Practice in the Best Management Practices Tool form.

To add the Best Management Practice specified in the drop down list, click the Add BMP button. The Best Management Practice for the selected drainage will be added to the Best Management Practice list.

*Create New Best Management Practices*

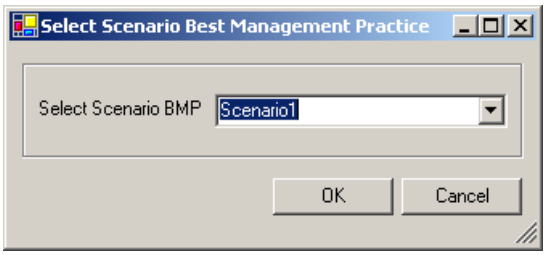
To create a new set of BMP's, click the button. You will be asked to confirm whether you would like to start with new BMP's, discarding any that may currently be displayed in the BMP list.




Click the Yes button to clear existing BMP's. Click the No button to keep the current list of BMP's.

*Opening Existing Best Management Practices*

To open a set of previously defined BMP click the button. The Select Scenario form will be displayed allowing the DSS user to select a set of BMP's associated with a DSS Scenario Run ID.



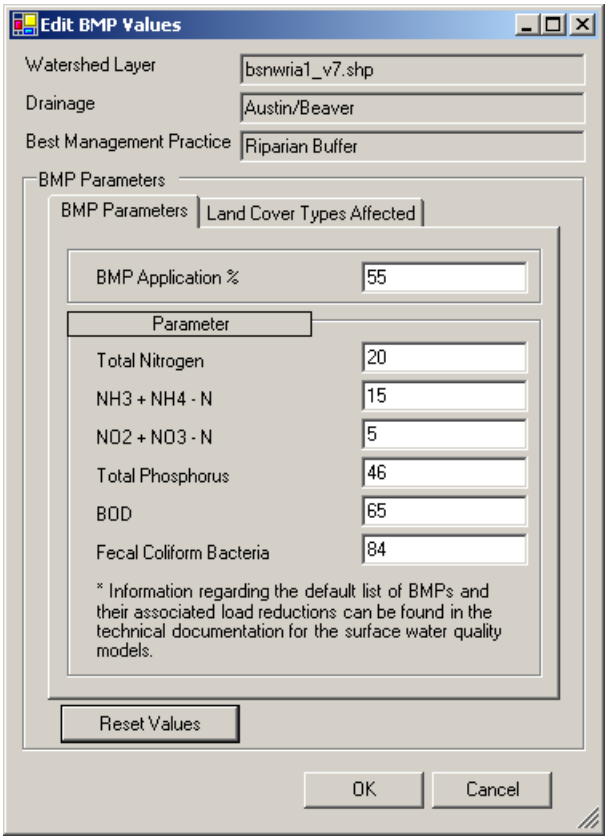
To open the BMP's contained in the selected scenario click the OK button. To cancel opening a set of BMP's from a previous scenario click the Cancel button.

*Save Best Management Practices*

To save the BMP's displayed in the BMP list click the button. This saves the BMP's displayed in the BMP list for the Water Quality Model in the current scenario layout.

*Edit Best Management Practices*

To edit the BMP selected in the BMP list click the button. The Edit BMP Values form will then be shown for the BMP selected in the BMP list.




To change a value for the BMP click on the BMP Parameters tab. To change a particular value for the BMP, enter a number between 0 and 100 into the text box to the right of the value you wish to edit. To change whether a land cover type is affected by the BMP, click on the Land Cover Types Affected tab. To change whether a particular land cover type is affected by the BMP check or uncheck the ☑ box to the left of the land cover type.

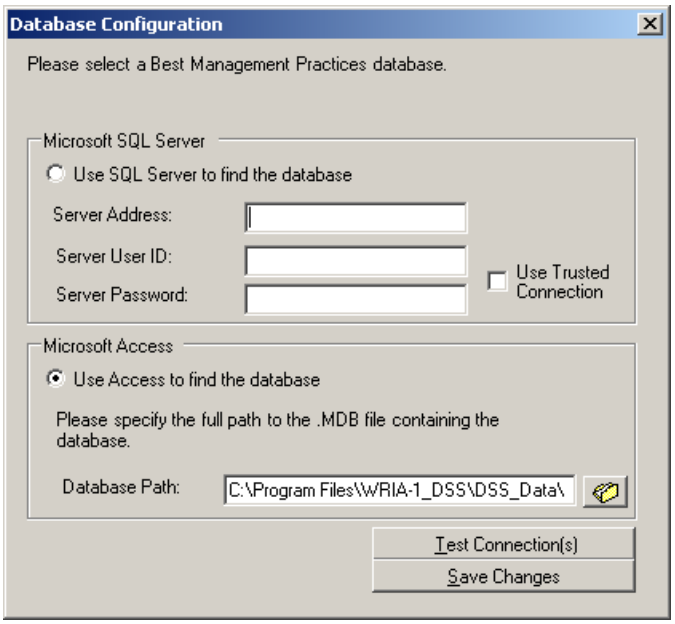
To use the changed BMP values click the OK button. To discard the changed BMP values click the Cancel button.

*Delete Best Management Practices*

To delete the BMP selected in the BMP list click the button. This removes the selected BMP from the BMP list.

*Change Best Management Practices Database*

To change the BMP database click the button. The Database Configuration form will be displayed, allowing the user to browse to the Best Management Practices Database.



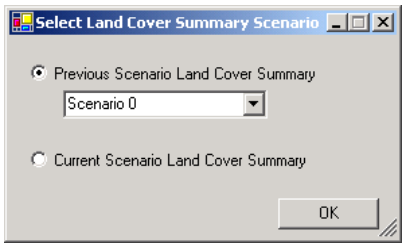

To browse to a Microsoft Access Best Management Practices Database click the button. An open file dialog will be displayed. The default location for the Best Management Practices is "C:\Program Files\WRIA-1_DSS\DSS_Data\Model Manager Databases\BestManagementPractices.mdb".

To save the selected database location, click the Save Changes button.

*Select Land Cover Summary Scenario*

The Select Scenario form allows the DSS user to select a DSS Scenario ID associated with a summary of land cover contained in the Land Cover Summary Database. To

access this form, select the [...] button to the right of **Land Cover Summary Scenario** on the Inputs tab of the Water Quality Model Inputs form. This will bring up the Land Cover Summary Scenario form as shown below.



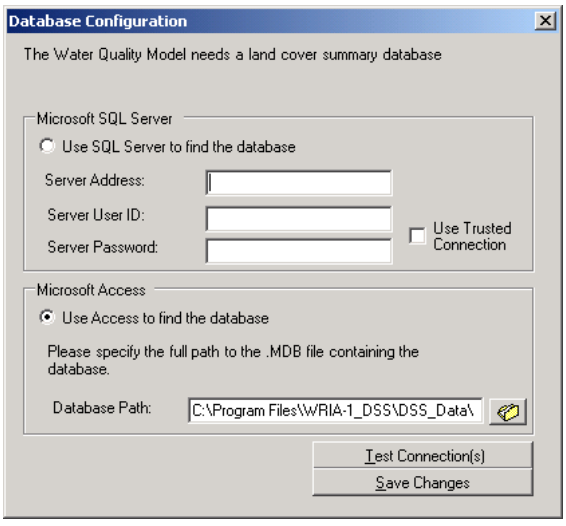

To select a previous scenario click the [Previous Scenario Land Cover Summary] radio button, then select a scenario from the drop down list.

To select the current DSS scenario (meaning the Land Cover Summary is contained in the same scenario layout as the Water Quality Model), click the [Current Scenario Land Cover Summary] radio button.

To save the scenario that has been selected, click the [OK] button.

*Select Land Cover Database*

The Database Configuration form allows a DSS user to select a Land Cover Summary. To access this form, select the [...] button to the right of **Land Cover Database Path** on the Input tab of the Water Quality Model Inputs form. This will bring up the Database Configuration form as shown below.

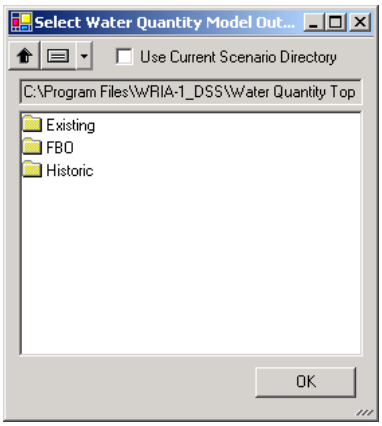

Database Configuration
The Water Quality Model needs a land cover summary database
Microsoft SQL Server
Use SQL Server to find the database
Server Address:
Server User ID:
Server Password:
Use Trusted Connection
Microsoft Access
Use Access to find the database
Please specify the full path to the .MDB file containing the database.
Database Path:
C:\Program Files\WRIA-1_DSS\DSS_Data\
Test Connection(s)
Save Changes

To browse to a Microsoft Access Land Cover Summary database click the button. An open file dialog will be displayed. The default location for the Land Cover Summary Database is "C:\Program Files\WRIA-1_DSS\DSS_Data\Model Manager Databases\LandCoverSummary.mdb".

To save the selected database location, click the Save Changes button.

*Select Water Quantity Output Directory*

The Select Directory form allows the DSS user to select a directory containing the Water Quantity Model output. To access this form, select the ... button to the right of **Water Quantity Output Directory** on Input tab of the Water Quality Model Inputs form. This will bring up the Select Water Quantity Model Output Directory form as shown below. The default location for the Water Quantity Model output is "C:\Program Files\WRIA-1_DSS\Water Quantity TopNet Files\Existing".

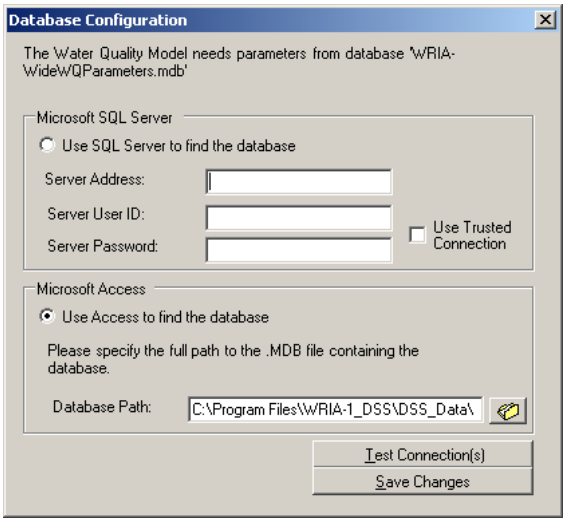



To move up a directory level click the button. To change the drive to browse for folders click the button. To use a directory that will be created when the current scenario is run, check the Use Current Scenario Directory box. To select a folder as the desired directory, click on the folder's name. To open a folder, double-click the folder's name.

To save the selected directory, click the OK button.

*Select Water Quality Parameters Database*

The Database Configuration form allows a DSS user to select a Water Quality Parameters Database. To access this form, select the ... button to the right of **Water Quality Parameters Database** on the Input tab of the Water Quality Model Inputs form. This will bring up the Database Configuration form as shown below.

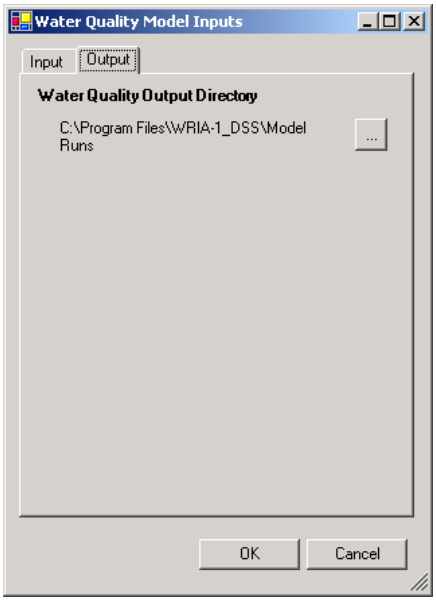



To browse to a Microsoft Access Water Quality Parameters database click the button. An open file dialog will be displayed. The default location for the WRIA-Wide Water Quality Parameters Database is "C:\Program Files\WRIA-1_DSS\DSS_Data\Model Manager Databases\WRIA-WideWQParameters.mdb".

To save the selected database location, click the Save Changes button.

### *Edit WRIA-Wide Water Quality Model Outputs*

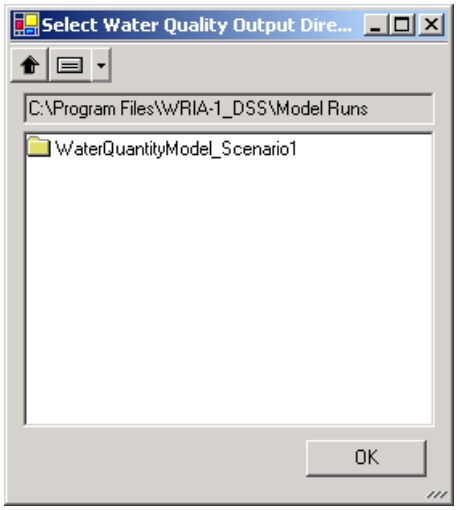
The Output tab allows the user to view/edit the output settings for the WRIA-Wide Water Quality Model. To change the settings for a particular output, select the ... button to the right of the output setting.



To save the changes to the settings that have been made in the outputs form, click the OK button. To cancel the changes to the settings click the Cancel button.

*Select Water Quality Output Directory*

The Select Directory form allows the DSS user to select a directory for the Water Quality Model output. To access this form, select the ... button to the right of **Water Quality Output Directory** on the Output tab of the Water Quality Model Inputs form. This will bring up the Select Water Quality Model Output Directory form as shown below. The default location for the Water Quality Model output is "C:\Program Files\WRIA-1_DSS\Model Runs".

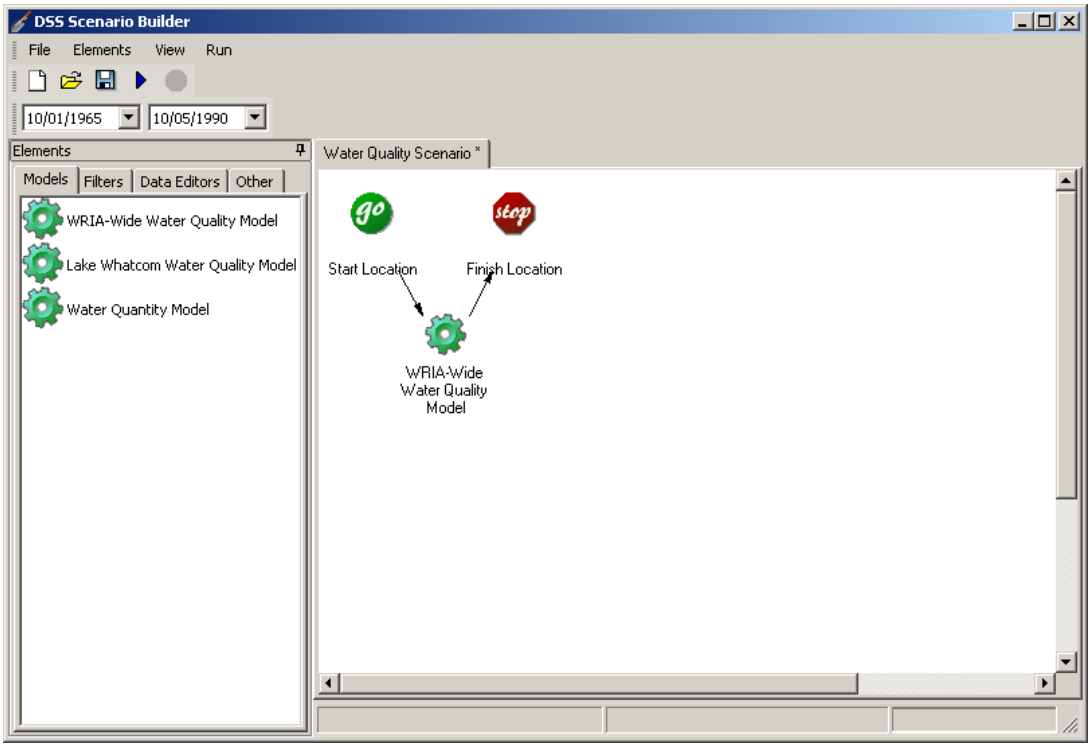



To move up a directory level click the button. To change the drive to browse for folders click the button. To select a folder as the desired directory, click on the folder's name. To open a folder, double-click the folder's name.

To save the selected directory, click the OK button.

## Run WRIA-Wide Water Quality Model

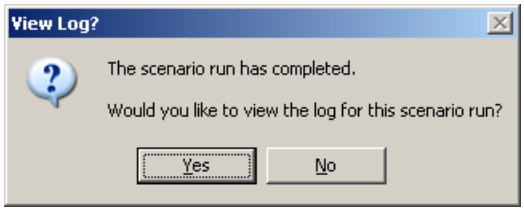
The DSS Scenario Builder allows the DSS user to run the Water Quality Model.

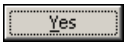
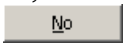



Before the Water Quality Model can be run, the start and end dates need to be specified in the Scenario Builder. To change the start and end dates, click . The start date is the date listed to the left, and the end date is the date listed on the right.

To run the scenario, click the button. If you would like to abort the scenario run, click the button to cancel the scenario run.

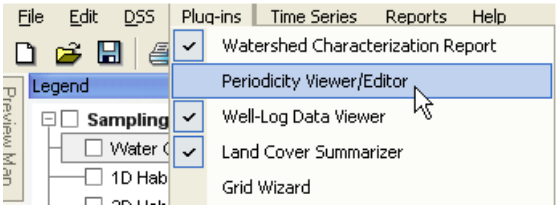
When the scenario has completed, the DSS user will be asked if they would like to view the scenario run log (a text file containing messages output by the elements contained in the scenario that has just been run).



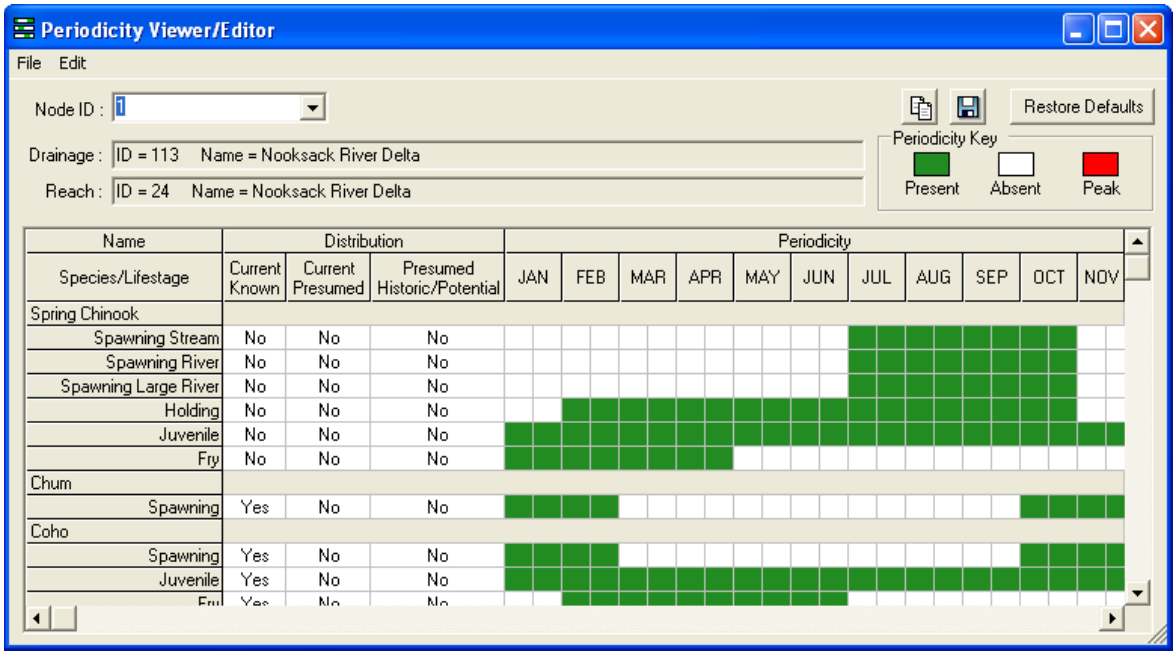

To view the log for the scenario run, click the  button. To decline viewing the scenario log at this time click the  button.

# Periodicity Viewer Plug-in

The Periodicity Viewer Plug-in to the DSS allows users to view and edit the species periodicity information used in the Habitat Time Series Model. To access this tool, press the Periodicity Viewer toolbar button on the DSS toolbar. If this button is not available on the toolbar, click on the Plug-ins menu on the menu bar and select Periodicity Viewer/Editor from the list as shown below.

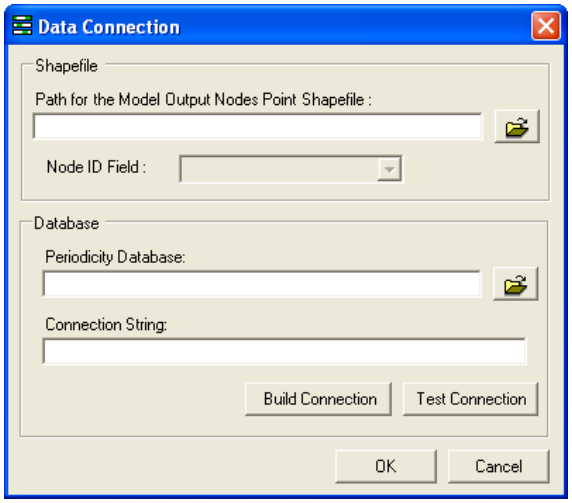



This will add the Periodicity Viewer button to the toolbar. When the toolbar button is clicked, the following form will appear.




## Configuring the Data Connection

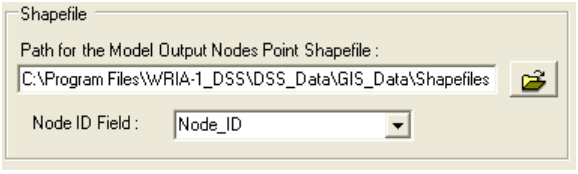
If the input and output data for the Periodicity Viewer/Editor has not been configured or has been configured incorrectly, the following screen will appear before the Periodicity Viewer/Editor will open, as shown below.




Before the Periodicity Viewer/Editor will run properly, the shapefile and database connections must be made.

### *Selecting the Model Output Nodes Point Shapefile*

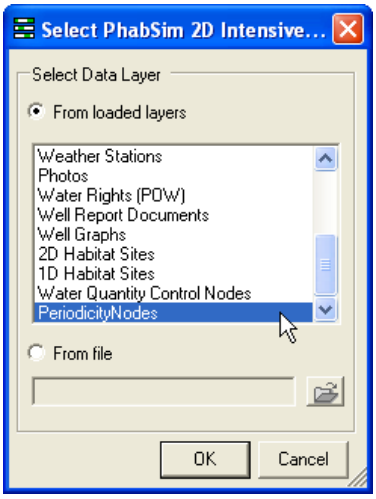
Fist, the path for the model output nodes point shapefile must be selected. This is done using the Shapefile section of the Data Connection form as shown below.




The default location for this shapefile is:

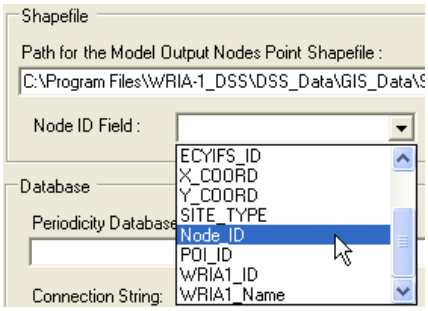
C:\ Program Files\WRIA-1_DSS\DSS_Data\GIS_Data\Shapefiles\ModelNodes\PeriodicityNodes.shp.

If this data was not installed in the default location, or you would like to use a different shapefile, click on the browse button to the right of the shapefile box. The following form will appear allowing the user to select the data layer from the loaded layers or from file.




If the Periodicity Nodes layer has been added to the current project, it will appear in the From loaded layers list as shown above. If it has not been added, select the From file button and press the browse button to navigate to the proper location. Once the desired shapefile has been selected, press OK to continue.

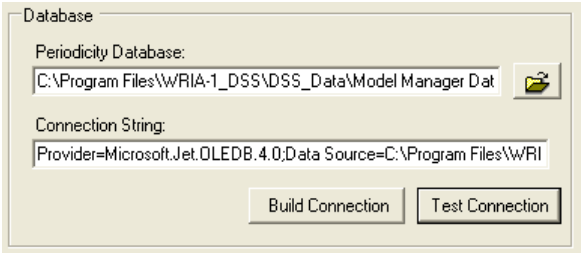
Once the shapefile has been selected, the Node ID field must be selected. Use the down arrow to the right of the Node ID filed box to select the desired field as shown below.




For the Periodicity Nodes shapefile the Node ID Field is Node_ID as shown above.

### *Selecting the Periodicity Database*

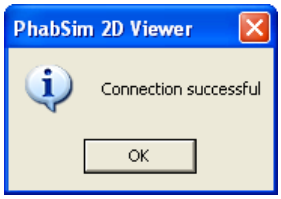
Once the path for the Model Output Nodes Point Shapefile has been successfully specified, the Periodicity Database connection must be made. This is done using the Database section of the Data Connection form as shown below.




The default location for the Periodicity database is given below:

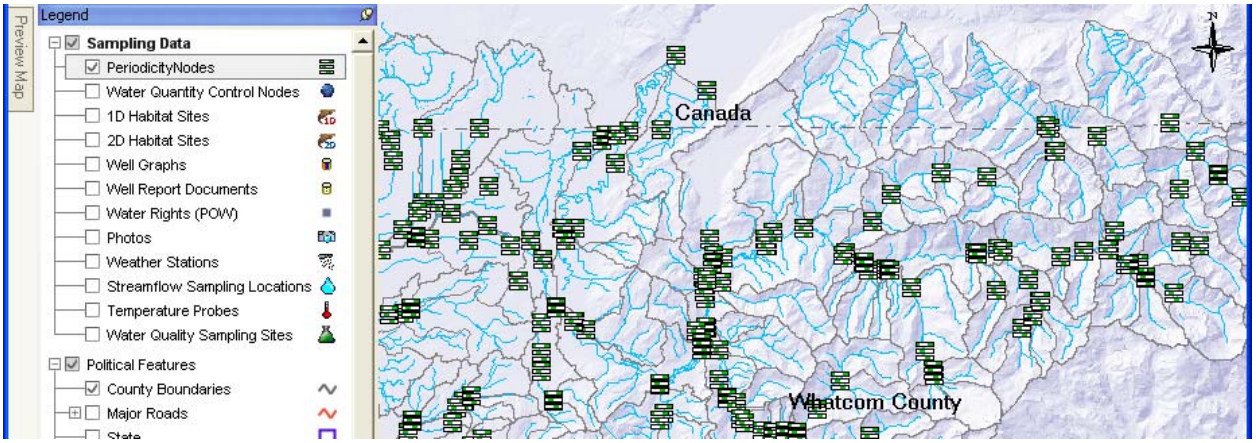
C:\Program Files\WRIA-1_DSS\DSS_Data\Model Manager Databases\mwPhabTSModel\Periodicity.mdb

If the data was not installed in the default location this link should be automatically updated for the correct location. If the link is not updated, or to select a different Periodicity Database, press the browse button to the right of the Periodicity Database pathfile box. Once a database has been selected, the connection string should be built automatically. If not, press the Build Connection button. Once the Connection String has been built, press the Test Connection button to test the connection. If the connection is successful, the following message will appear.




After a successful periodicity database connection has been made and the proper shapefile and Node ID have been selected, press OK on the Data Connection form to continue.

The Periodicity Nodes will be added to the layers and displayed on the map, similar to the image shown below.

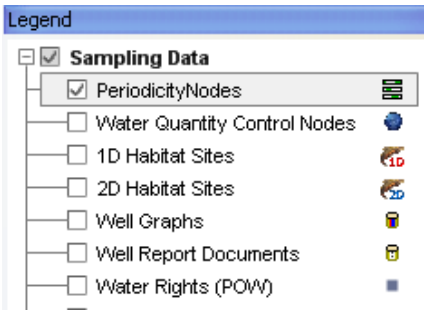


## Viewing Periodicity Data

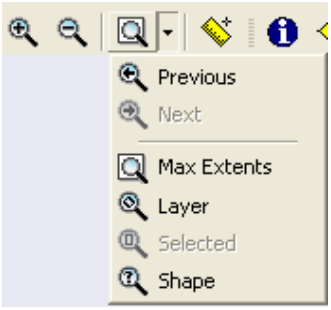
Periodicity data can be viewed by either clicking on and selecting a Periodicity Node point or group of points from the map or by clicking on the Periodicity Viewer toolbar button. Selecting points from the map loads the available periodicity data for the selected locations into the Periodicity Viewer/Editor first, followed by all the other nodes in numerical order. Clicking on the Periodicity Viewer toolbar button loads all available periodicity data in the entire database into the Periodicity Viewer/Editor in numerical order.

### *Selecting Photo Viewer Points from the Map*

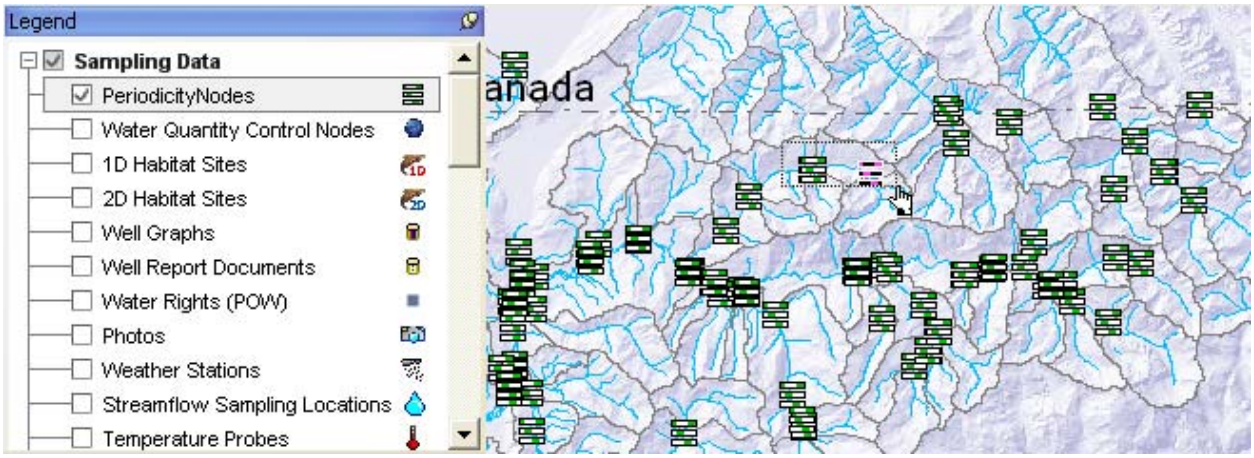
Make sure that the Periodicity Nodes theme is turned on (checked) and active (selected).



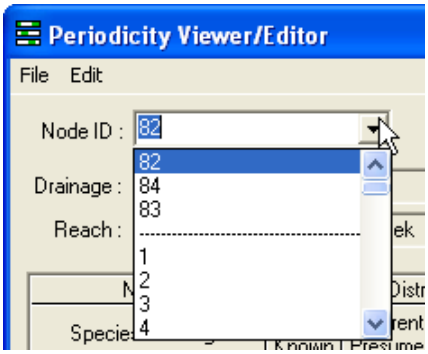

Next, use the Zoom and Pan Tools to locate the area of interest on the map. Additional Zoom tools are accessed by clicking on the down arrow as shown below.




Using the select tool , points can be selected by either clicking on individual stations or by clicking and dragging a box around one or several locations as shown in the example below.




Selecting points from the Periodicity Nodes points theme on the map will load the periodicity data for those selected locations into the Periodicity Viewer/Editor form, followed by the rest of the periodicity data in the database in numerical order. The example below shows Nodes 82, 84 and 83 loaded into the Periodicity Viewer/Editor after being selected from the map.




***Viewing Periodicity Data via the Periodicity Viewer Toolbar Button***

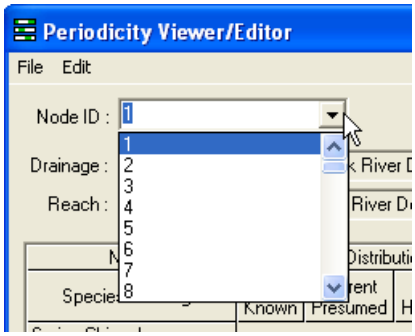
As an alternative to selecting points using the map, clicking the Periodicity Viewer toolbar button opens the Periodicity Viewer/Editor and loads the entire database of periodicity data into the Periodicity Viewer/Editor form. By pressing this button, users can browse the entire database of periodicity data without interacting with the map.

## Periodicity Viewer/Editor Options

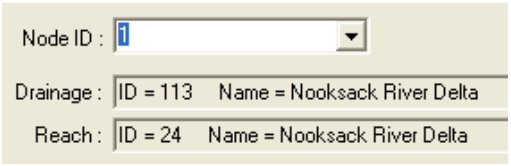
Several options are available on the Periodicity Viewer/Editor form. This form is quite large and is best viewed when maximized. The various options are described in the following sections.

### *Node Selection*

Users can select from the list of available locations by using the Node ID pull down box near the top right hand section of the Periodicity Viewer/Editor form as shown below.




Once a Node ID has been selected, the drainage and reach information will be displayed in the boxes as shown below.

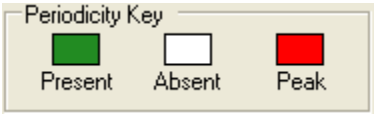



Below the drainage and reach information, the periodicity data is displayed for the selected node.

| Name | Distribution | | | Periodicity | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Species/Lifestage | Current Known | Current Presumed | Presumed Historic/Potential | JAN | FEB | MAR | APR | MAY | JUN | JUL | AUG | SEP | OCT | NOV | DEC |
| Spring Chinook | | | | | | | | | | | | | | | |
| Spawning Stream | No | No | No | | | | | | | ■ | ■ | ■ | ■ | ■ | ■ |
| Spawning River | No | No | No | | | | | | | ■ | ■ | ■ | ■ | ■ | ■ |
| Spawning Large River | No | No | No | | | | | | | ■ | ■ | ■ | ■ | ■ | ■ |
| Holding | No | No | No | | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ |
| Juvenile | No | No | No | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ |
| Fry | No | No | No | ■ | ■ | ■ | ■ | ■ | ■ | | | | | | |

### *Viewing Distribution and Periodicity Data*

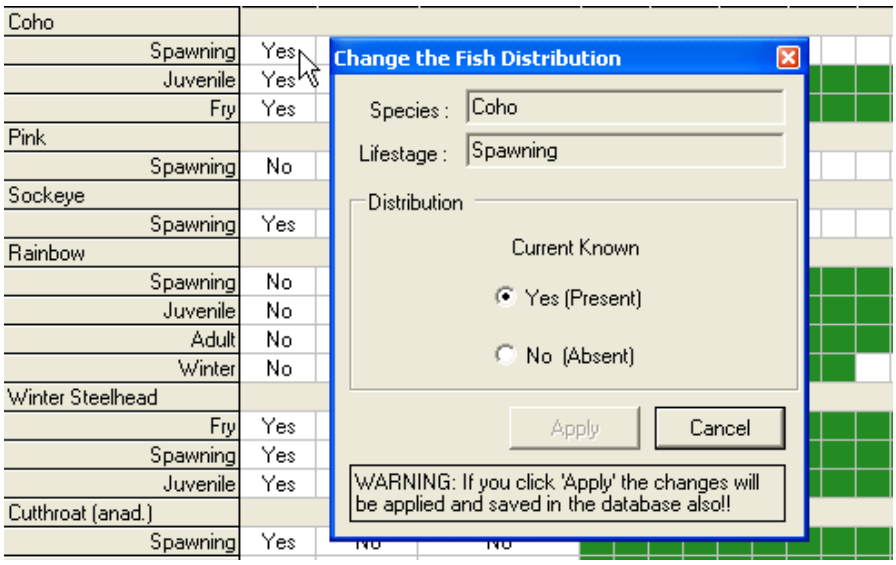
The first, left- most column displays the names of the species and lifestages. This example shows Spring Chinook and its six lifestages. The second column shows the distribution data for each species and lifestage. There are three distributions, Current Know, Current Presumed, and Presumed Historic/Potential. The third and right- most column displays the bi-monthly periodicity data for each species and lifestage. Notice that there are two boxes beneath each month. The first box represents the first half of that month, and the second is the second half. A key for the periodicity data is shown near the right hand corner of the Periodicity Viewer/Editor form as shown below.




Green represents Present, white represents Absent, and Red represents Peak. Periodicity data is given for 43 species and lifestages. To view the data, use the scroll bar located along the right side of the Periodicity Viewer/Editor form.

### *Editing Distribution Data*

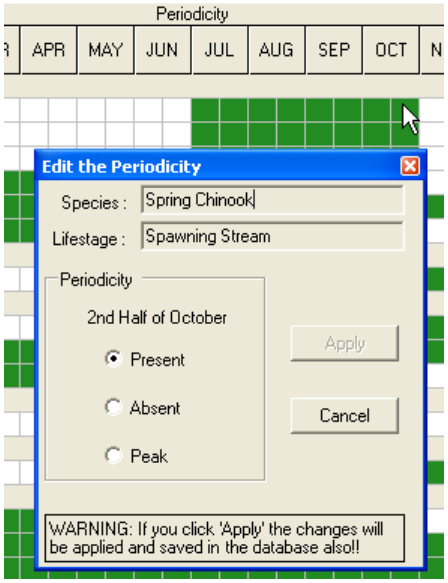
To edit the Distribution Data for a given species and lifestage at the selected Node, double-click on the box representing the desired lifestage and distribution. This will bring up the Change Fish Distribution form as shown below. The example below shows the form for Coho Spawning, Current Known distribution.




To change the distribution value, click on the radio buttons in the Distribution section of the Change Fish Distribution form next to Yes (Present) or No (Absent). To apply the change and save it to the database, press Apply. Note that pressing this button will apply the changes to the on screen view, and will also change the Distribution database. To return to the Periodicity Viewer/Editor form without changing the distribution data, press Cancel.

### *Editing Periodicity Data*

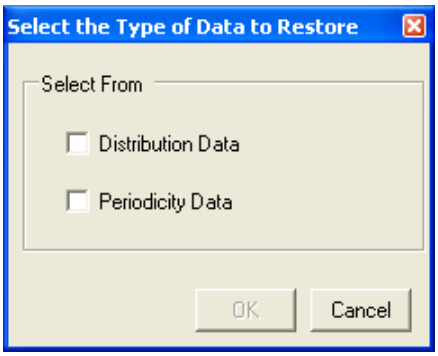
To edit Periodicity Data for a given species and lifestage at the selected Node, double-click on the box representing the desired time period and species. This will bring up the Edit Periodicity Data form as shown below. The example below shows the Edit Periodicity form for the second half of October for species: Spring Chinook and lifestage: Spawning Stream.




Using the radio buttons in the Periodicity section of the form, the selected periodicity value can be changed to Present, Absent, or Peak. To change the data, press Apply. Note that pressing this button will apply the changes to the on screen view, and will also change the Periodicity database as well. To return to the Periodicity Viewer/Editor form without changing the periodicity data, press Cancel.

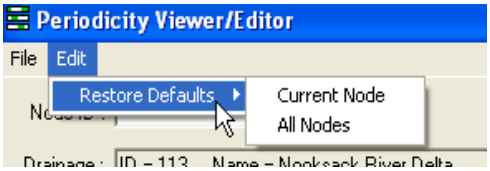
***Restoring Database Values***

To restore the default values to the database, press the Restore Defaults button located in the top right-hand corner of the Periodicity Viewer/Editor form. This will bring up the following box, allowing the user to choose to restore the Periodicity database, the Distribution database or both.




This will restore the data for all the nodes in the database.

Default values can be restored for the Current Node or All Nodes by clicking on the **Edit** menu, and selecting either **Current Node** or **All Nodes** as shown below.

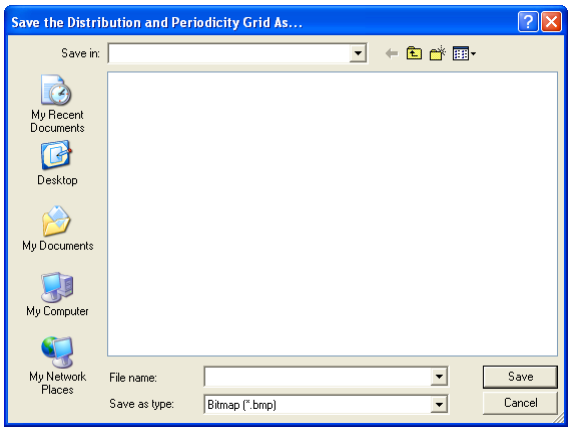



This will again bring up the Select the Type of Data to Restore box allowing the user to restore data from the Periodicity and/or Distribution databases.

### *Copy/Save the Distribution and Periodicity Grid*

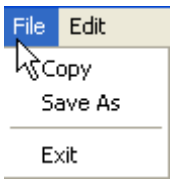
To copy the Distribution and Periodicity Grid as shown on the screen, press the copy button located near the top right-hand corner of the Periodicity Viewer/Editor form. This will copy an image of the Distribution and Periodicity Grid to the clipboard and allow it to be pasted into documents. Also, the grid may be copied by clicking on the **File** menu and selecting **Copy**.

To save the Distribution and Periodicity Grid, press the save button located near the top right-hand corner of the Periodicity Viewer/Editor form. This will open a save dialog box as shown below, allowing the user to select a destination and filename for the grid.




The grid will be saved as a bitmap.

Also, the grid may be copied or saved by clicking on the **File** menu and selecting **Copy** or **Save As** as shown below.

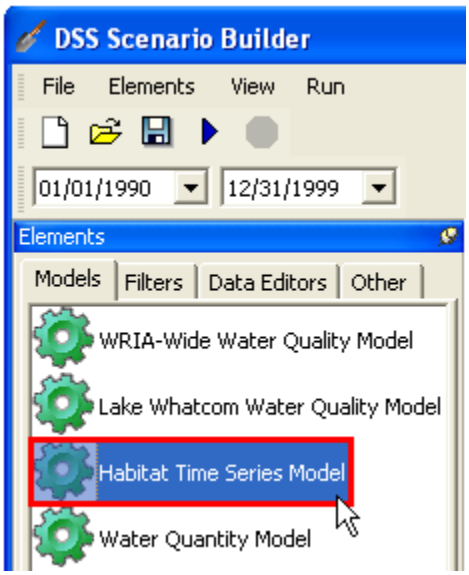


# Habitat Time Series Model Interface

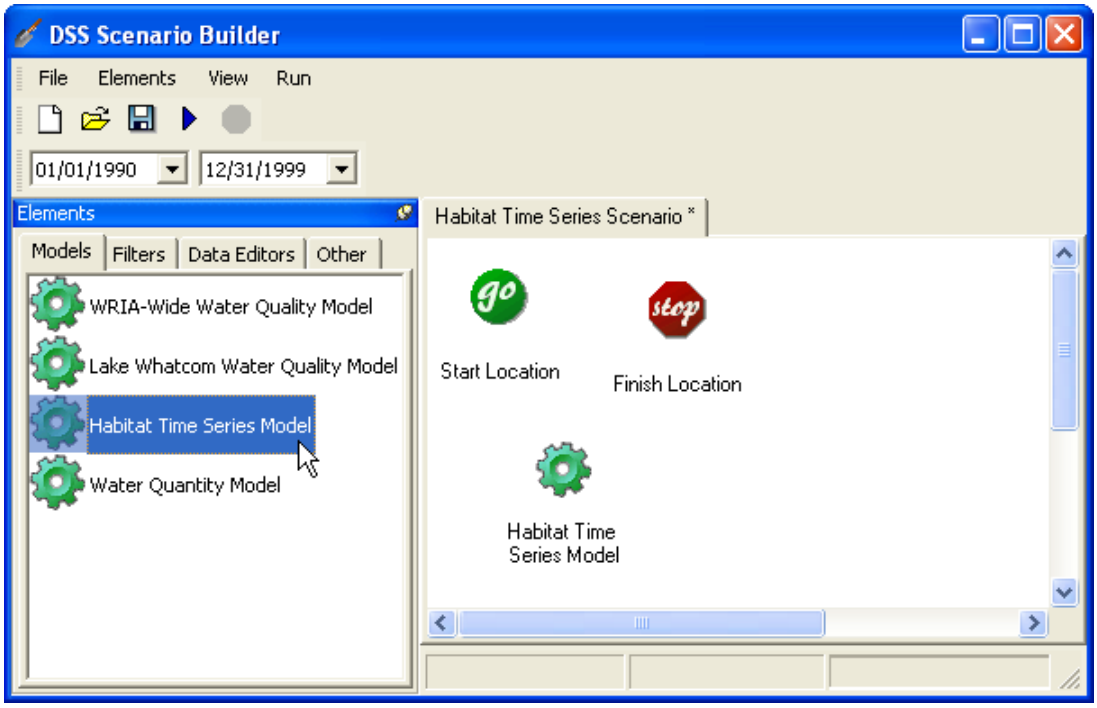
The Habitat Time Series Model Interface allows the DSS user to specify the input for the Water Quality Model for the WRIA-1 Drainages.

## Add Habitat Time Series Model to DSS Scenario

To add the Habitat Time Series Model to a DSS scenario layout, first open the DSS Scenario Builder by choosing the Scenario Builder option under the DSS menu. Create a new scenario layout by clicking . Click on the Models tab in the Elements window of the Scenario Builder and double click on the Habitat Time Series Model highlighted in red below.

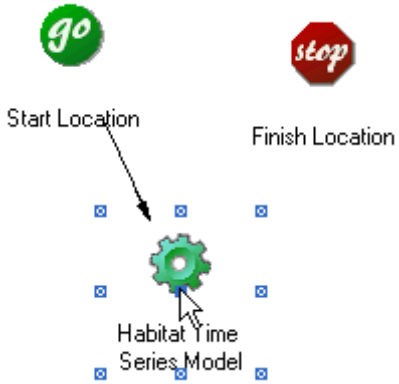


The Habitat Time Series Model will be added to the scenario layout as shown below.

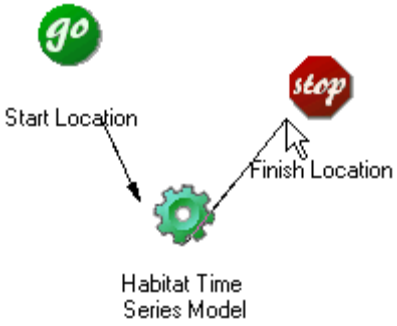



To prepare the Habitat Time Series Model to be run in the Scenario Builder, you must connect the model to the Start Location and the Finish Location nodes in the scenario layout.

First, click on the Start Location node. Then, click and hold the mouse on the box in the center of the node's icon. While still pressing the mouse button, drag the mouse to the Habitat Time Series Model node in the scenario layout.

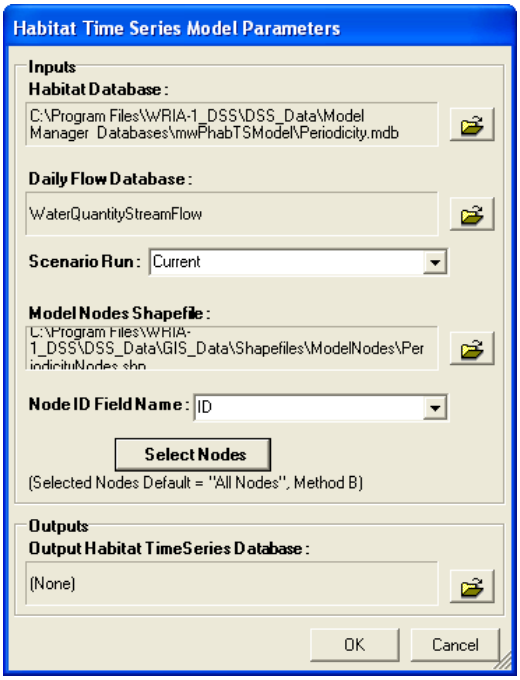



Repeat the previous step by again clicking the box in the center of the Habitat Time Series Model, dragging the mouse to the Finish Location node.

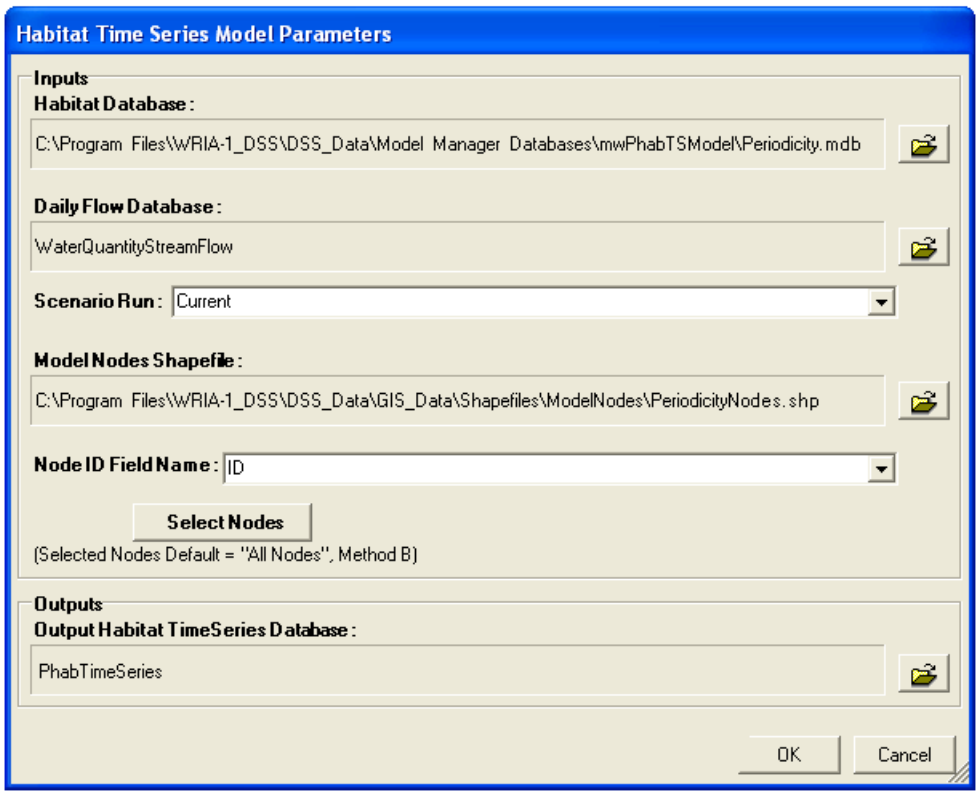



The Habitat Time Series Model is now connected properly to be run in the DSS Scenario Builder. The model will not run successfully without the proper settings. To edit the Habitat Time Series Model settings, double click on the Habitat Time Series Model node in the scenario layout as shown below.

The Habitat Time Series Model settings form will be displayed as seen below.




The size of this from can be increased by clicking on the bottom right hand corner of the form and dragging. This is useful when attempting to view the pathnames for the various input files. The form below has been expanded to show the entire pathname of all the files.

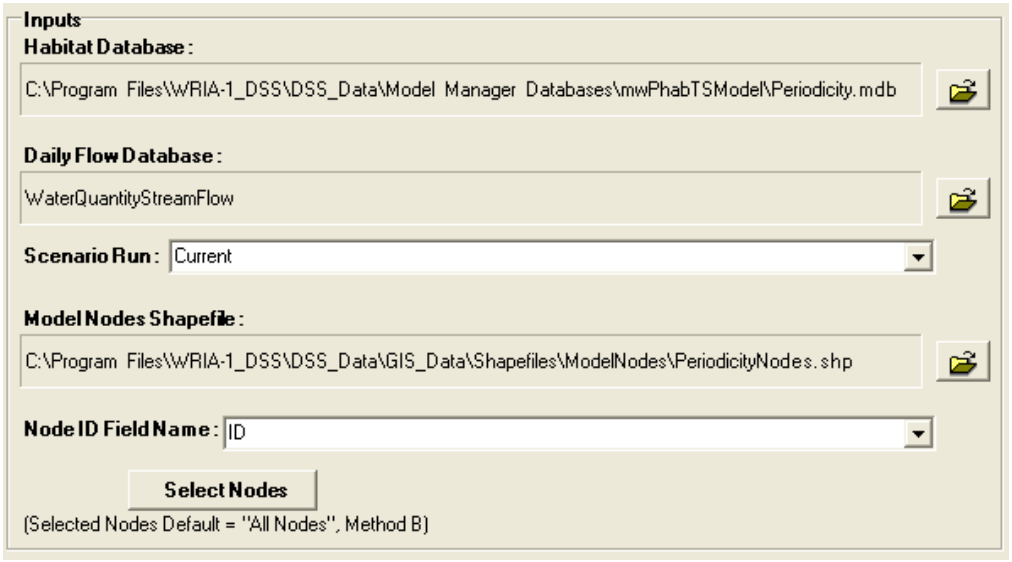


## Edit Habitat Time Series Model Settings

The inputs for the Habitat Time Series Model include the following: Habitat Database, Daily Flow Database, Scenario Run, Model Nodes Shapefile, Node ID Field Name, and Select Nodes. Each of these options may be edited in the Inputs section of the Habitat Time Series Model Parameters form. How to edit each of these options is discussed in the following section titled Edit Habitat Time Series Model Inputs. The output options for the Habitat Time Series Model consist of selecting and output database. The output database can be selected using the Outputs section of the Habitat Time Series Model Parameters form. More information about this process is given in the section titled Edit Habitat Time Series Model Outputs.

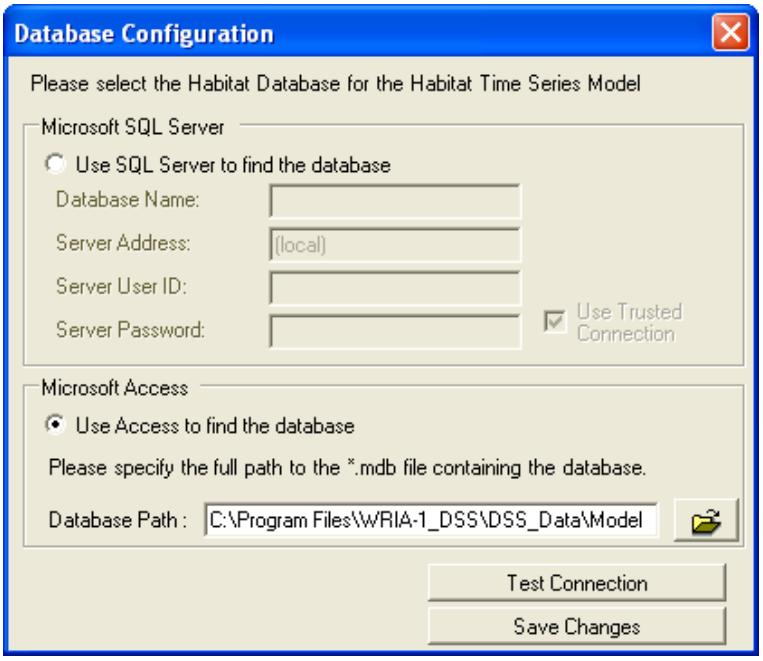
### *Edit Habitat Time Series Model Inputs*

The Inputs section of the Habitat Time Series Model Parameters form allows the DSS user to edit the inputs for the Habitat Time Series Model.

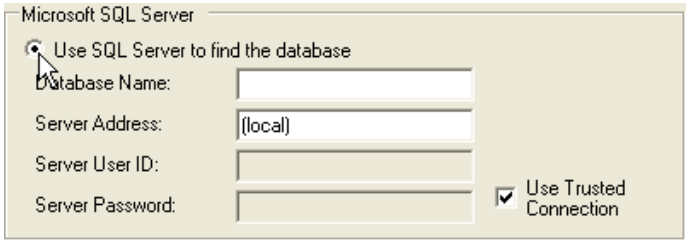



*Select Habitat Database*

The first option listed in the Inputs section allows the user to select the Habitat Database used. The default location for this database is listed in the form above. To select a different database, press the browse button to the right of the Habitat Database pathname. This will bring up the Database Configuration form as shown below.




The user has the option of selecting a Microsoft SQL Server database by selecting the Use SQL Server to find the database option. The user must then enter the database information in the following boxes.

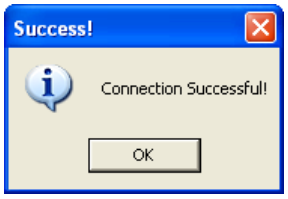



To use a Microsoft Access database as the input Habitat Database, the user can select the Use Access to find the database option.

Microsoft Access
Use Access to find the database
Please specify the full path to the *.mdb file containing the database.
Database Path :
C:\Program Files\WRIA-1_DSS\DSS_Data\Model

The user must then enter the complete pathname in the box to the right of Database Path: or use the browse button to navigate to the desired database.

To test the connection to the database and to verify that the database is available, press the Test Connection button. If a successful connection is made to the database listed the following message will appear.

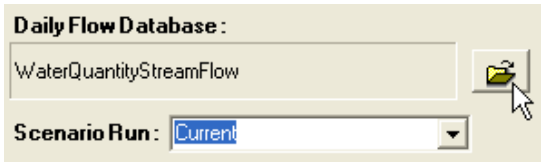



If the database cannot be found, the following error will be displayed.

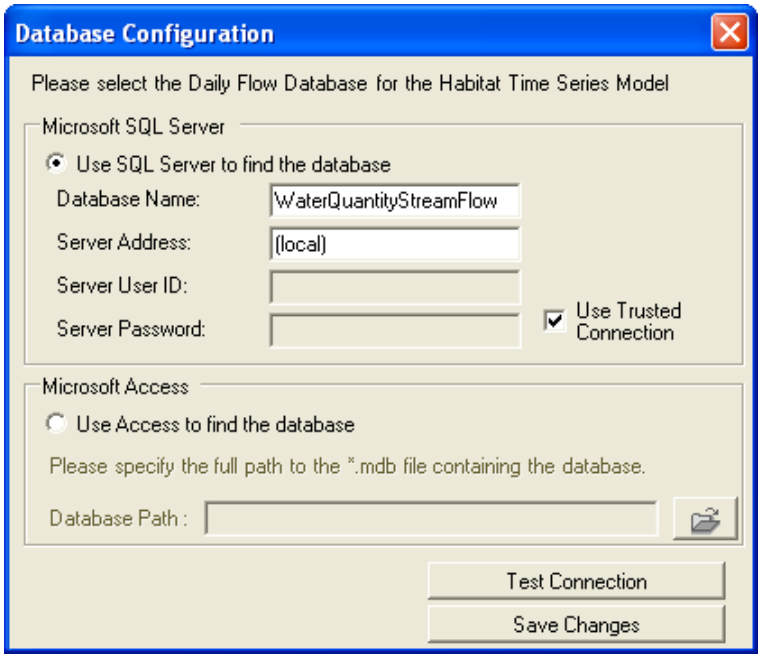
Once a successful database connection has been made to the desired database, press the Save Changes button to continue.

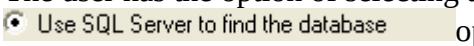
*Select Daily Flow Database*

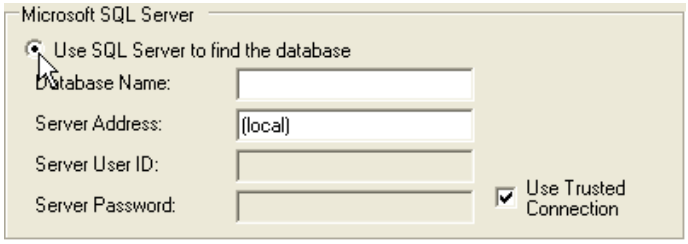
The default database for the Daily Flow Database is the WaterQuantityStreamFlow database. To select a different database, click the browse button to the right of the database name as shown below.

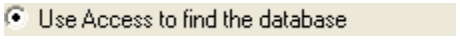



This will bring up the Database Configuration form.




The user has the option of selecting a Microsoft SQL Server database by selecting the  option. The user must then enter the database information in the following boxes.

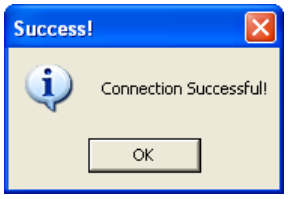



To use a Microsoft Access database as the input Daily Flow Database, the user can select the  option.

Microsoft Access
Use Access to find the database
Please specify the full path to the *.mdb file containing the database.
Database Path :
C:\Program Files\WRIA-1_DSS\DSS_Data\Model

The user must then enter the complete pathname in the box to the right of Database Path: or use the browse button to navigate to the desired database.

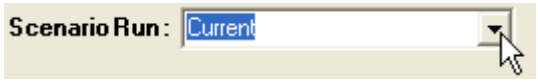
To test the connection to the database and to verify that the database is available, press the Test Connection button. If a successful connection is made to the database listed the following message will appear.




If the database cannot be found, the following error will be displayed.

Once a successful database connection has been made to the desired database, press the Save Changes button to continue.

Once a database has been selected, the user must specify which Water Quantity scenario in the Daily Flow Database the Habitat Time Series Model should use. This is done by clicking the down arrow to the right of the Scenario Run: box as shown below.



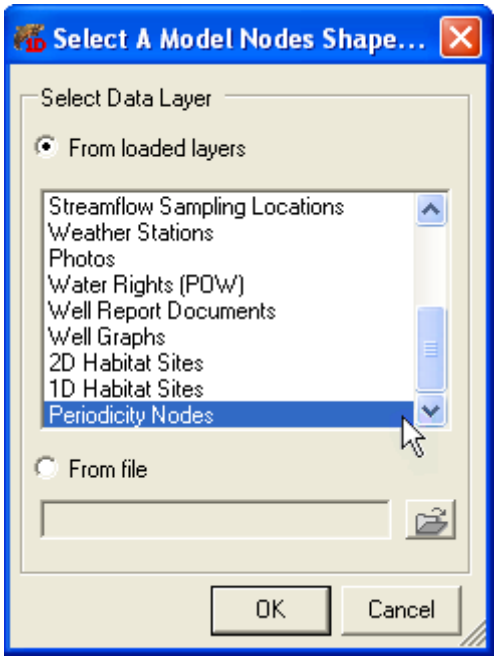

The user may select any of the Water Quantity scenarios contained in the Daily Flow Database, or choose to use the data from the Water Quantity model contained in the current scenario. Thus, the Current option only works when the current scenario contains a Water Quantity model.

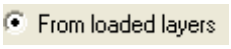
*Select Model Nodes Shapefile*

The default Model Nodes Shapefile location is shown below.

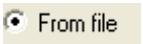
To change the Model Nodes Shapefile used in the Habitat Time Series Model, click the browse button to the right of the Model Nodes Shapefile pathname. The following

form will appear, allowing the user to select a different shapefile from the loaded layers or from file.

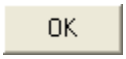


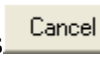

To select a Model Nodes Shapefile from the layers currently loaded in the DSS, select the 

option and select a layer from the list as shown above.

To select a Model Nodes Shapefile from file, select the 

option and enter the entire file pathname in the box below or press the browse button to navigate to the desired file location.

Once a layer or file has been selected, press 

to accept the shapefile and continue. To return to the Habitat Time Series Model Parameters form without selecting a new Model Nodes shapefile, press 

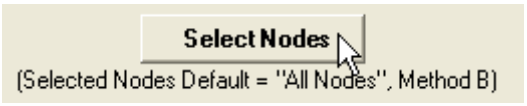
.

The user must specify which field in the selected Model Nodes shapefile is to be used using the Node ID FieldName box as shown below.

If more than one Field Name exists in the selected Model Nodes shapefile, it can be selected from the list by pressing the down arrow to the right of the Node ID Field Name box as shown above.

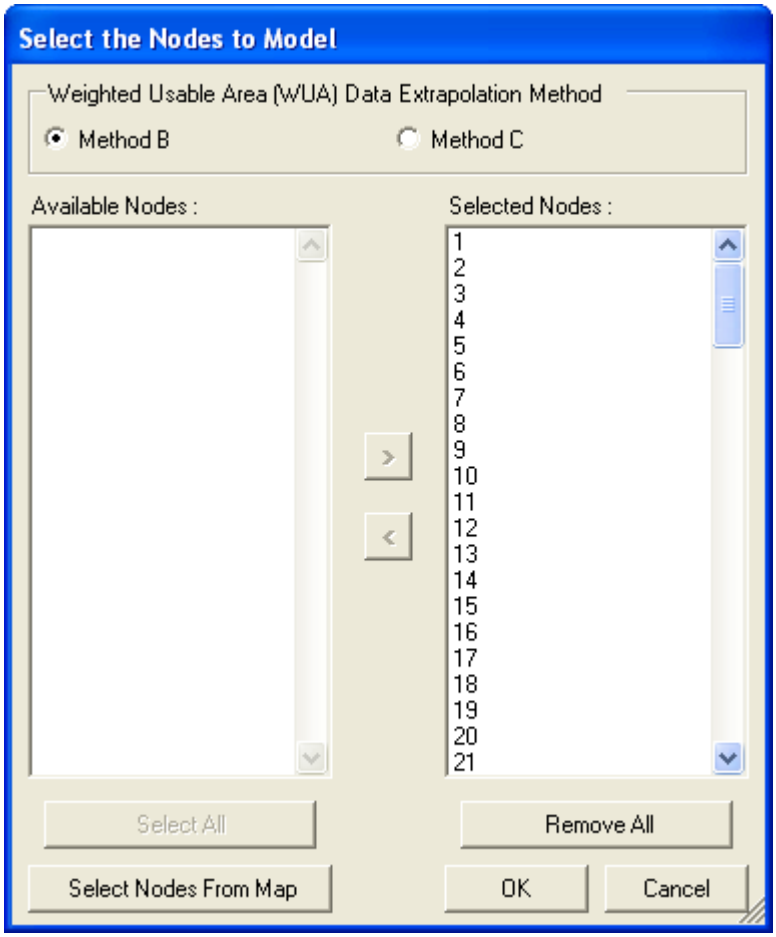
*Select Nodes*

To select the Nodes to be modeled in the Habitat Time Series Model, press the **Select Nodes** button as shown below.




Note that if this button is not pushed, the model will use the default of all nodes selected and will use WUA calculation method B.

Pressing the **Select Nodes** button brings up the following form.

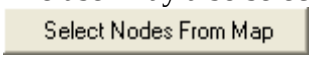


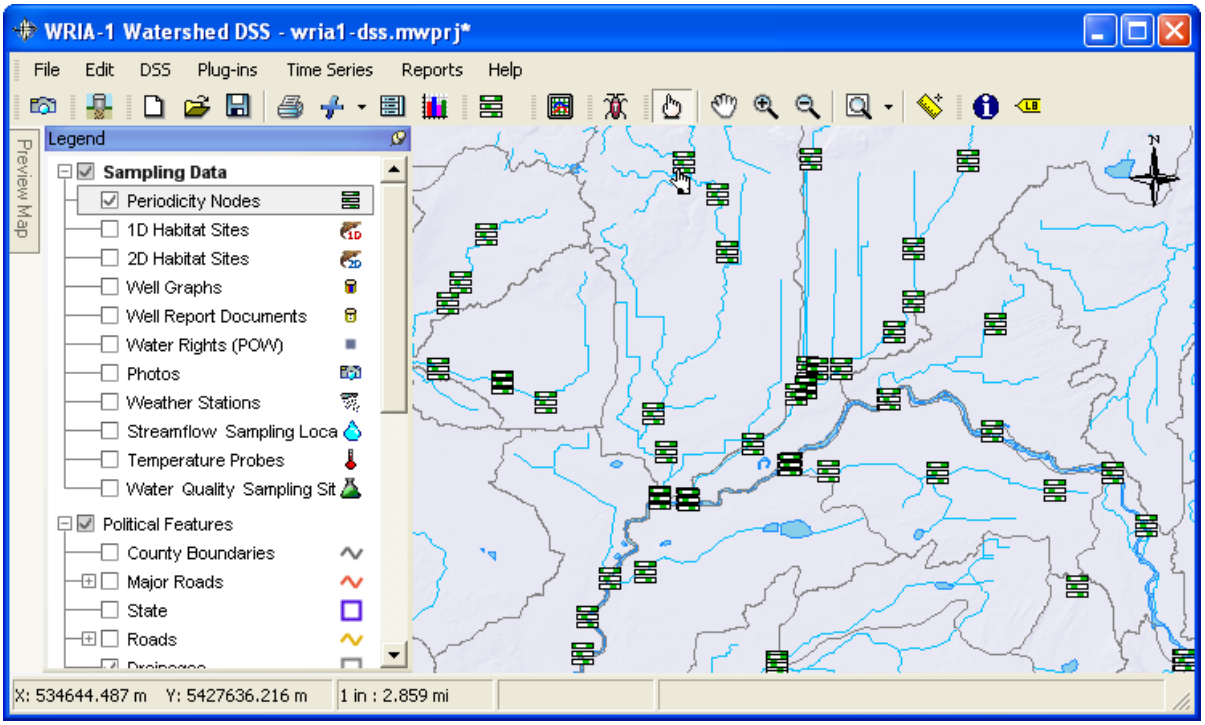
Again, the default is to model all the nodes and to use Method B. The user can select either WUA data extrapolation method B or C by selecting the radio button next to the desired extrapolation method.

Since by default, all the nodes are selected, they are listed in numerical order on the right hand side of the form in the Selected Nodes window. To remove a node from the Selected Nodes window, click on the node number with the mouse to highlight that node then, press the remove button. This will remove the selected node from the Selected Nodes window. This node will now be available and will be listed in the Available Nodes window on the left hand side of the form. To remove all the nodes in the Selected Nodes window, press the button. This will remove all the nodes from the Selected Nodes window and they will be displayed in the Available Nodes window.

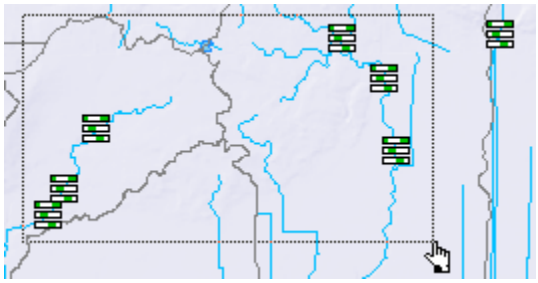
To select a node from the Available Nodes, click on the node number with the mouse to highlight that node, then press the add button to move the selected node to the Selected Node window.

Multiple nodes can be selected for addition or removal by holding down the Ctrl key while using the mouse to select the desired nodes.

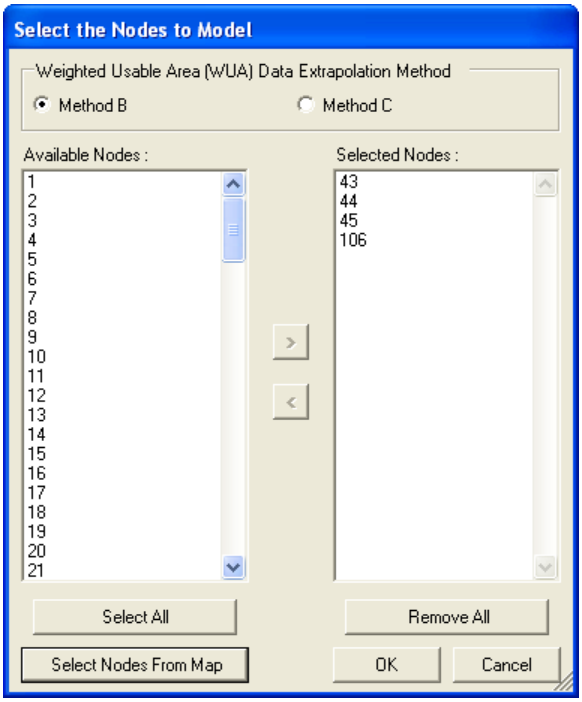
The user may also select specific nodes from the map by pressing the  button. This will minimize all other windows allowing the user to view the map. To select a node from the map, the desired node shapefile layer must be active and selected. This layer must be the same shapefile entered in the Model Nodes Shapefile section described above. If the user attempts to select nodes from a shapefile other than the one entered in the Model Nodes Shapefile section, and error will occur. To select nodes, the user must use the Select Shapes tool. In the example shown below, nodes from the Periodicity Nodes layer are being selected.



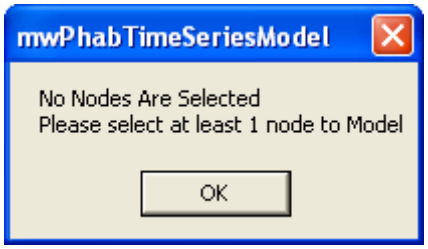

Once a node has been selected, the Select the Nodes to Model form will reappear, and the selected node will be listed in the Selection Nodes section of the form. Additional nodes can be selected by pressing the Select Nodes From Map button and clicking on another node. The user may also select multiple nodes at once by clicking and dragging the mouse to create a rectangle around the desired nodes. This will select all nodes that lie within the rectangle as shown below.



The four selected nodes have now been added to the Selected Nodes section of the form as shown below.



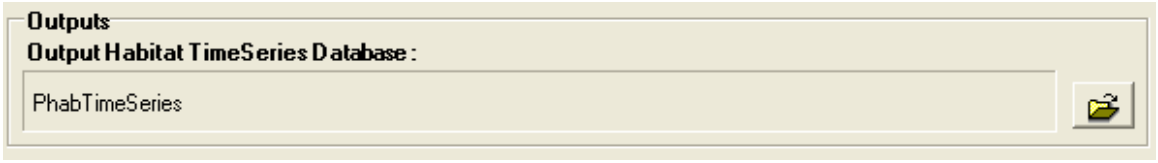

Once all the desired nodes have been selected and are listed in the Selected Nodes box, and the desired extrapolation method has been selected, press OK to accept the selections and return to the Habitat Time Series Model Parameters form. The user must select at least one node to model or the following error will occur.




To return to the Habitat Time Series Model Parameters form from the Select the Nodes to Model form without selecting any nodes or an extrapolation method, press the Cancel button. The default options will then be used to model.

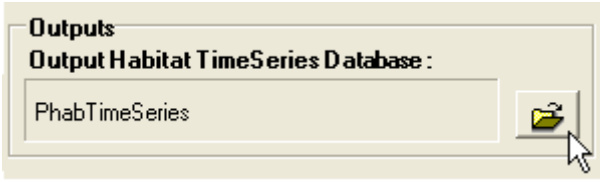
### *Edit Habitat Time Series Model Outputs*

The Outputs section of the Habitat Time Series Model Parameters form, located near the bottom of the form, allows the DSS user to edit the outputs for the Habitat Time Series Model.

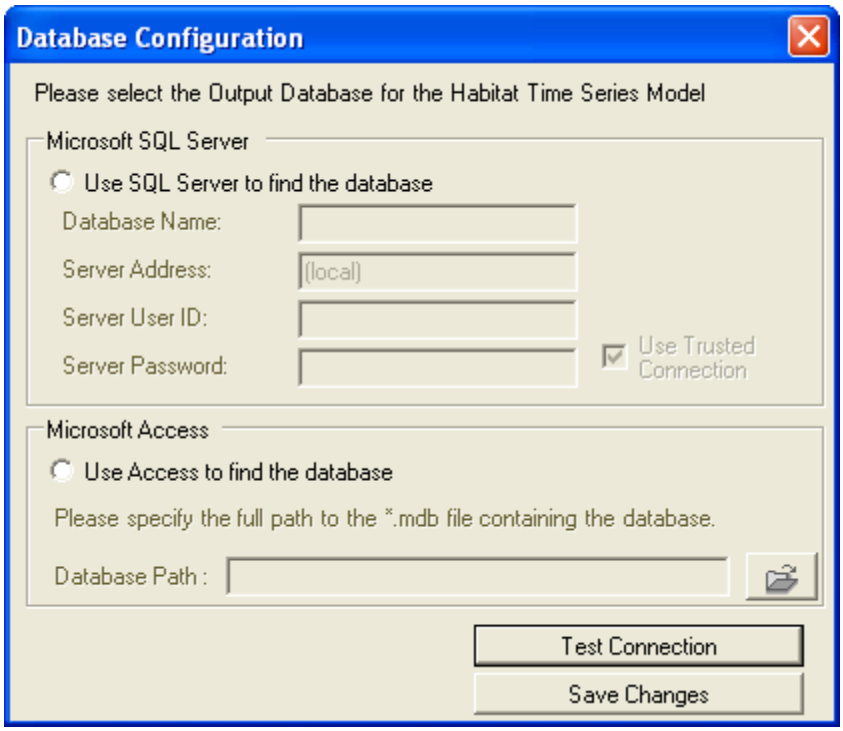



*Select Habitat Time Series Output Database*

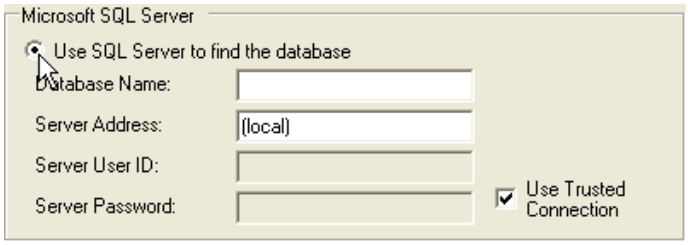
The default output database for the Habitat Time Series Model is the PhabTimeSeries database. To select a different database, click the browse button to the right of the database name as shown below.




This will bring up the Database Configuration form.




The user has the option of selecting a Microsoft SQL Server database by selecting the option. The user must then enter the database information in the following boxes.

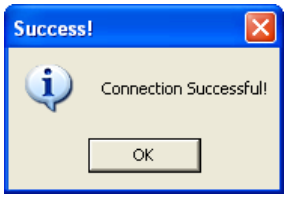



To use a Microsoft Access database as the Output Habitat Time Series Database, the user can select the Use Access to find the database option.

Microsoft Access
Use Access to find the database
Please specify the full path to the *.mdb file containing the database.
Database Path :
C:\Program Files\WRIA-1_DSS\DSS_Data\Model

The user must then enter the complete pathname in the box to the right of Database Path: or use the browse button to navigate to the desired database.

To test the connection to the database and to verify that the database is available, press the Test Connection button. If a successful connection is made to the database listed the following message will appear.




If the database cannot be found, the following error will be displayed.

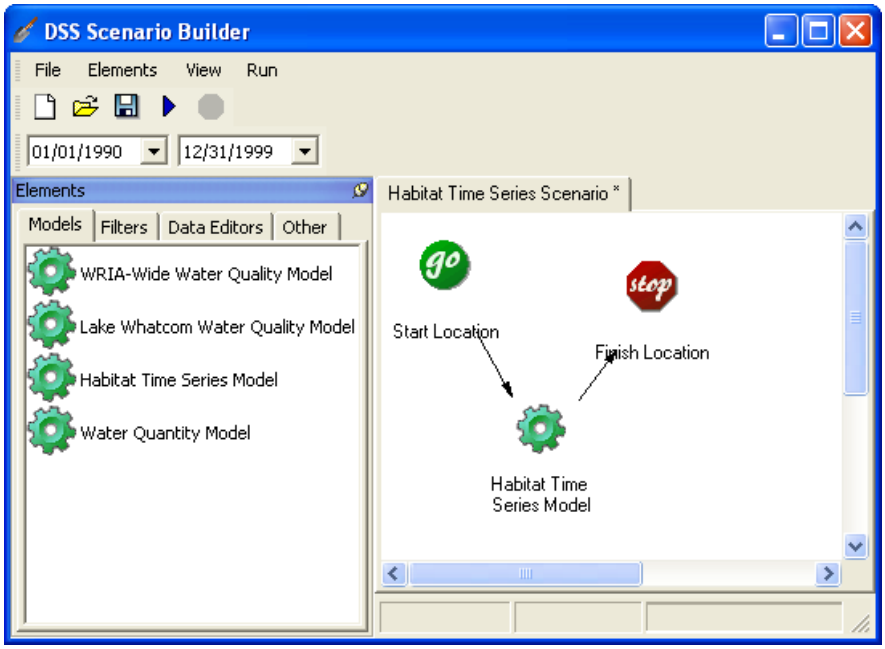
Once a successful database connection has been made to the desired database, press the Save Changes button to continue.

Once all the input and output parameters have been set using the Habitat Time Series Model Parameters form, press OK to accept the changes and return to the DSS

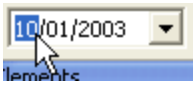
Scenario Builder window. To return to this window without accepting the changes made to the Habitat Time Series Model Parameters form, press Cancel.

## Run Habitat Time Series Model

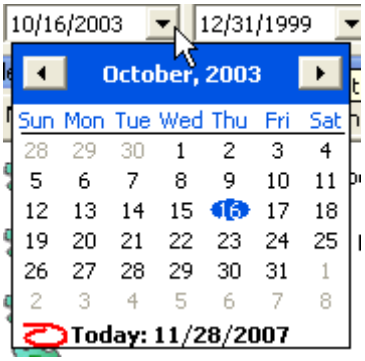
The DSS Scenario Builder allows the DSS user to run the Habitat Time Series Model.




Before the Habitat Time Series Model can be run, the start and end dates need to be specified in the Scenario Builder. To change the start and end dates, click on the date boxes 10/01/1965 10/05/1990 located below the toolbar near the left hand corner of the window. The start date is the date listed to the left, and the end date is the date listed on the right. The dates can be edited by clicking on the number in the box and typing in the desired values as shown below.

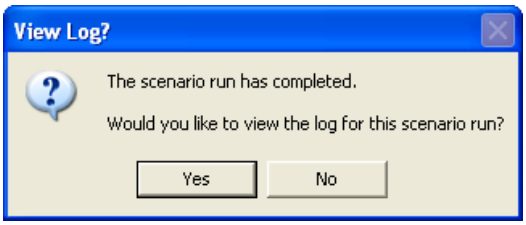



Or by clicking on the down arrow to the right of the date and selecting the desired start and end dates from the calendar as shown below.




Once the appropriate dates have been selected the model can be run. To run the scenario, click the button. If you would like to abort the scenario run, click the button to cancel the scenario run.

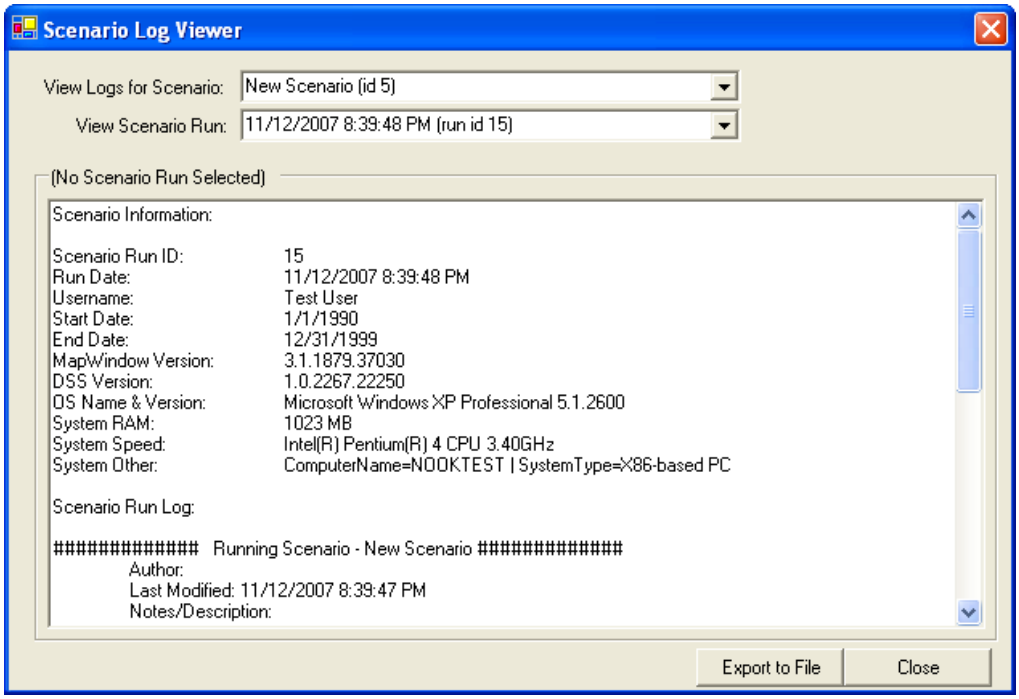
When the scenario has completed, the DSS user will be asked if they would like to view the scenario run log (a text file containing messages output by the elements contained in the scenario that has just been run).




To view the log for the scenario run, click the Yes button. To decline viewing the scenario log at this time click the No button.

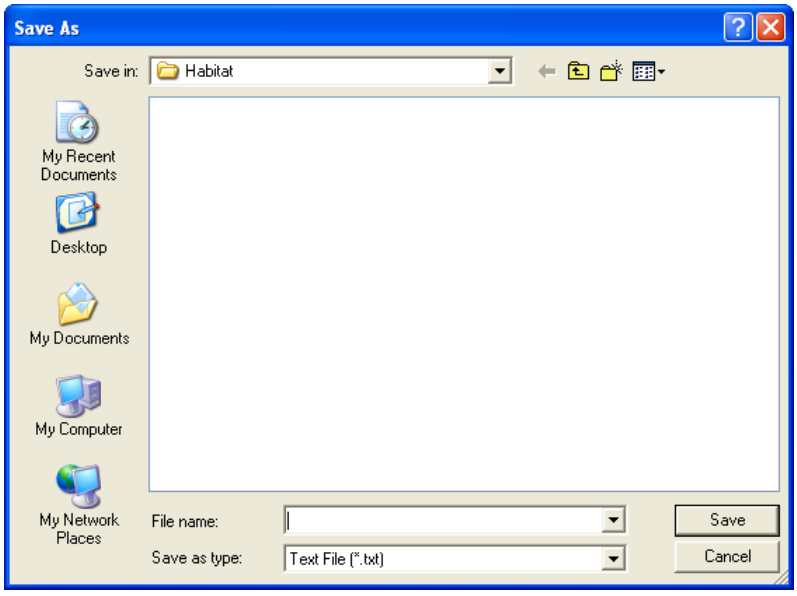
The scenario run log is saved for each individual execution of the scenario. The log contains information about the user running the scenario, their computer, and the elements involved in the scenario. Each element that runs contributes a segment to the run log, including details about input and output files and what actions the element took.

An example scenario log viewer is shown below:




### *Exporting Scenario Run Log*

The scenario run log may be exported, which is useful for archiving scenario results. Exporting the log may be done by clicking the "Export to File" button in the scenario log viewer (above). You will be asked where to save the file:



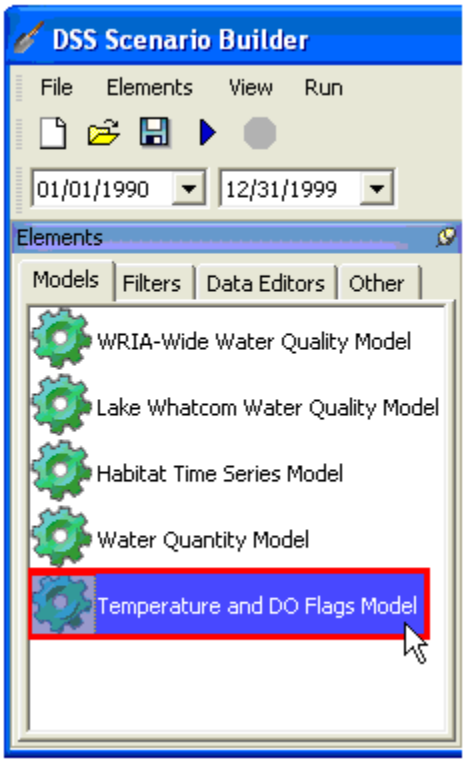

The output file will be textual (ASCII text), and will contain exactly what the scenario log viewer shows.

# Temperature and Dissolved Oxygen Flags Model Interface

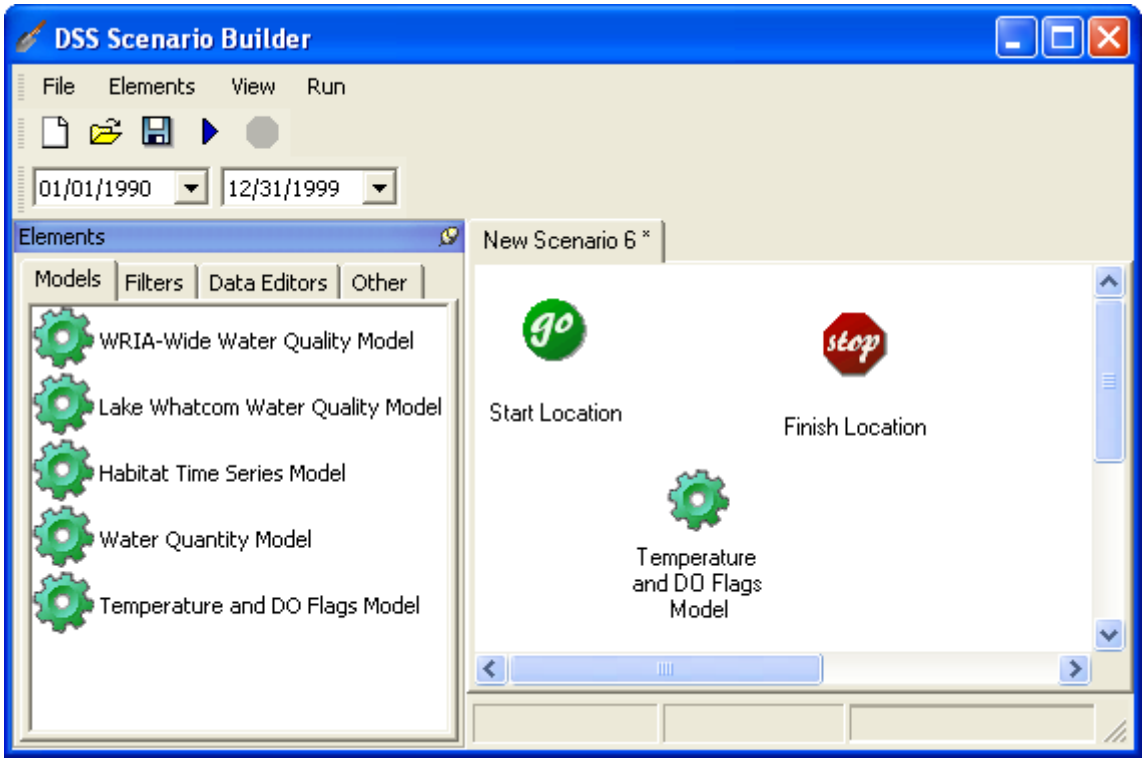
The Temperature and Dissolved Oxygen (DO) Flags Model Interface allows the DSS user to specify the input for the Temperature and DO Flags Model.

## Add Temperature and DO Flags Model to DSS Scenario

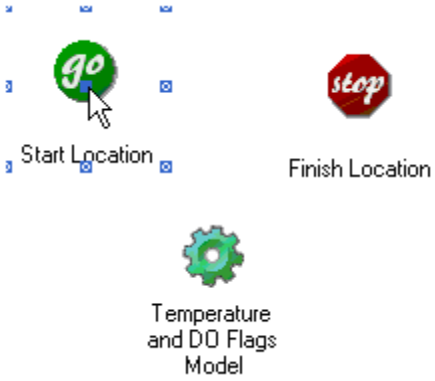
To add the Temperature and Dissolved Oxygen Flags Model to a DSS scenario layout, first open the DSS Scenario Builder by choosing the Scenario Builder option under the DSS menu. Create a new scenario layout by clicking . Click on the Models tab in the Elements window of the Scenario Builder and double click on the Temperature and DO Flags Model highlighted in red below.



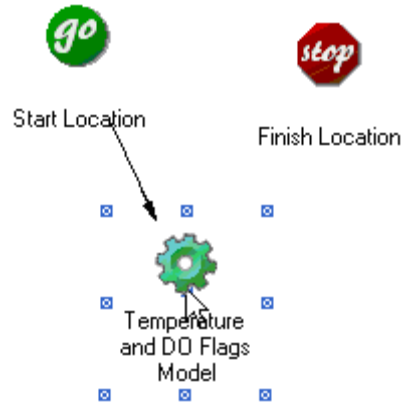
The Temperature and DO Flags Model will be added to the scenario layout as shown below.



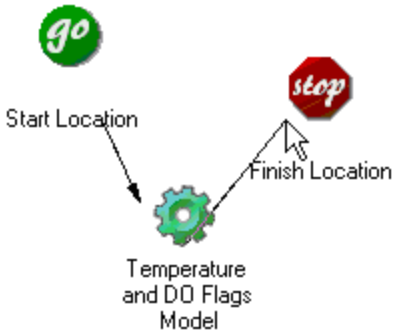

To prepare the Temperature and DO Flags Model to be run in the Scenario Builder, you must connect the model to the Start Location and the Finish Location nodes in the scenario layout.



First, click on the Start Location node. Then, click and hold the mouse on the box in the center of the node's icon. While still pressing the mouse button, drag the mouse to the Temperature and DO Flags Model node in the scenario layout.

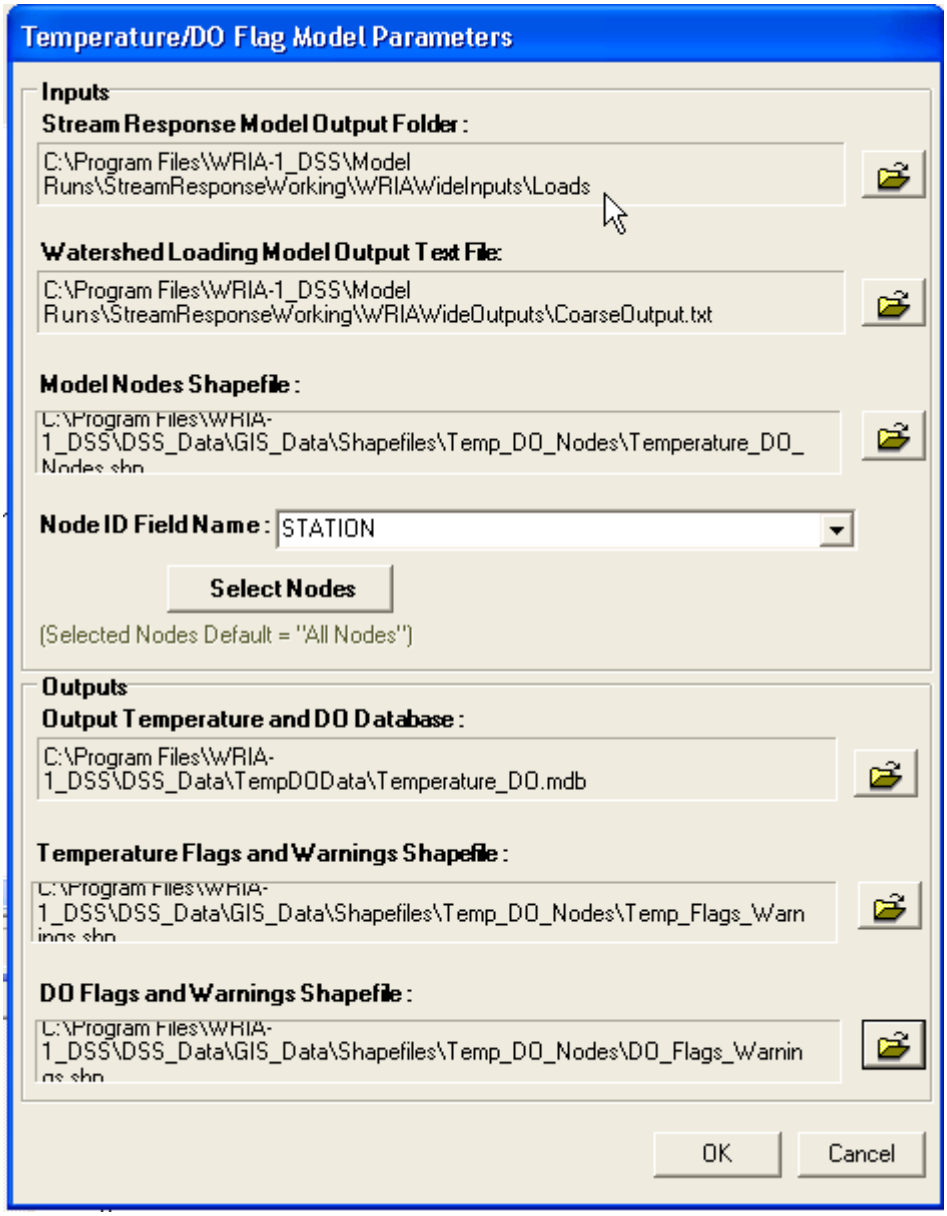



Repeat the previous step by again clicking the box in the center of the Temperature and DO Flags Model, dragging the mouse to the Finish Location node.




The Temperature and DO Flags Model is now connected properly to be run in the DSS Scenario Builder. The model will not run successfully without the proper settings. To edit the Temperature and DO Flags Model settings, double click on the Temperature and DO Flags Model node in the scenario layout as shown below.

The Temperature and DO Flags Model settings form will be displayed as seen below.



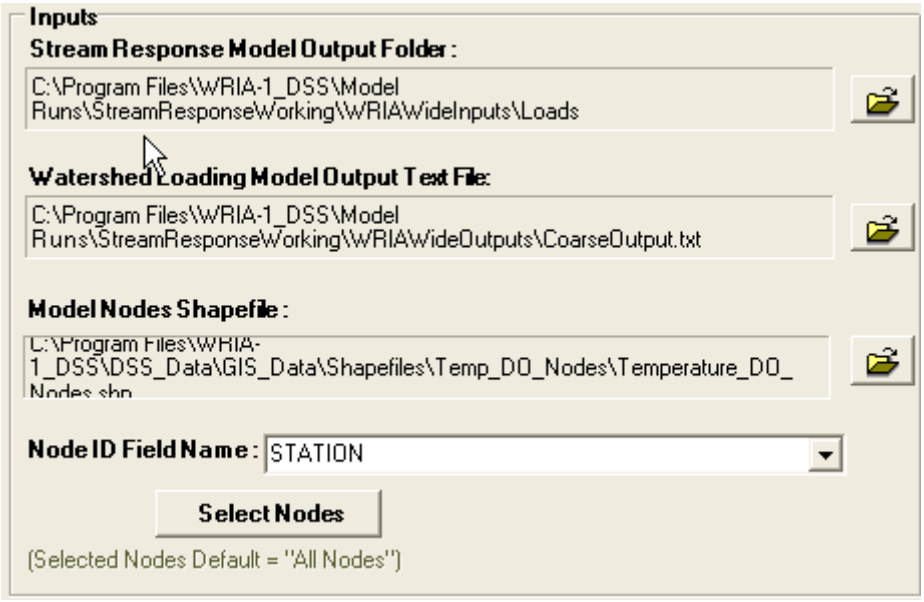

## Edit Temperature and DO Flags Model Settings

The inputs for the Temperature and DO Flags Model include the following: Stream Response Model Outputs Folder, Watershed Loading Model Output Text File, Model Nodes Shapefile, Node ID Field Name and Select Nodes. Each of these options may be edited in the Inputs section of the Temperature and DO Flags Model Parameters form. How to edit each of these options is discussed in the following section titled Edit Temperature and DO Flags Model Inputs. The output options for the Temperature and DO Flags Model consist of selecting the Output Temperature and DO Database,

Temperature Flags and Warnings Shapefile and DO Flags and Warning Shapefile. These outputs can be selected using the Outputs section of the Temperature and DO Flags Model Parameters form. More information about this process is given in the section titled Edit Temperature and DO Flags Model Outputs.

### *Edit Temperature and DO Flags Model Inputs*

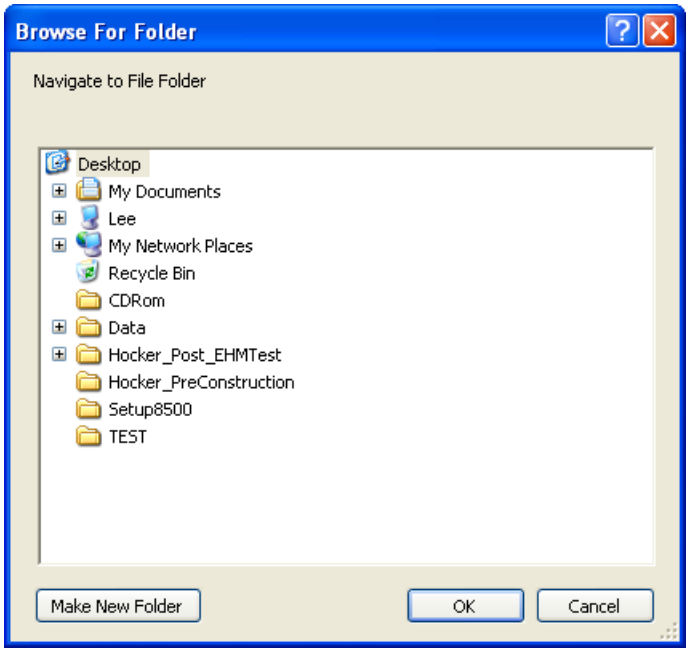
The Inputs section of the Temperature and DO Flags Model Parameters form allows the DSS user to edit the inputs for the Temperature and DO Flags Model.




*Select Stream Response Model Output Folder*

The first option listed in the Inputs section allows the user to select the foleder containing the output text files from the waterbody response model. The text files in this folder must be named NodeTS###.txt where the ### represents the node number of the output file.

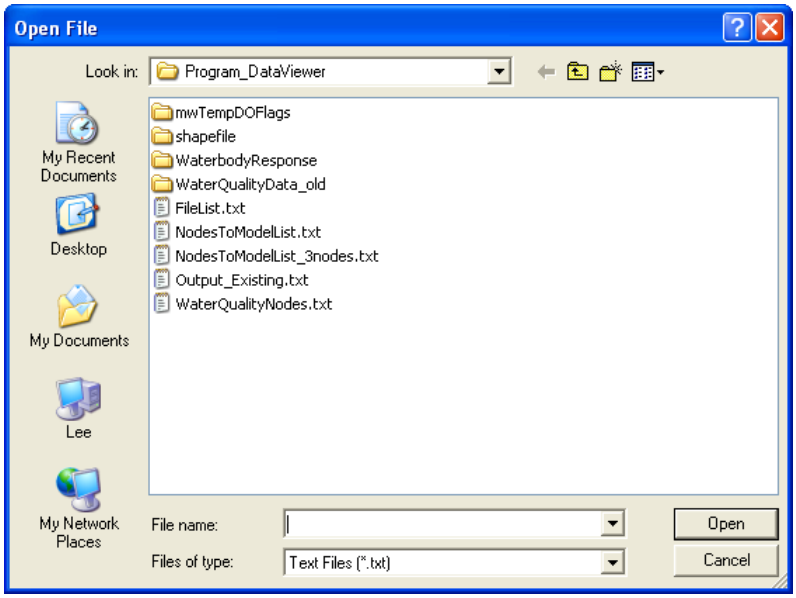
Habitat Database used. Press the browse button to the right of the Stream Response Model Output Folder pathname. This will bring up the Browse for Folder form as shown below.




Select the correct folder containing the waterbody response model output files and press OK.

*Select Watershed Loading Output Text File*

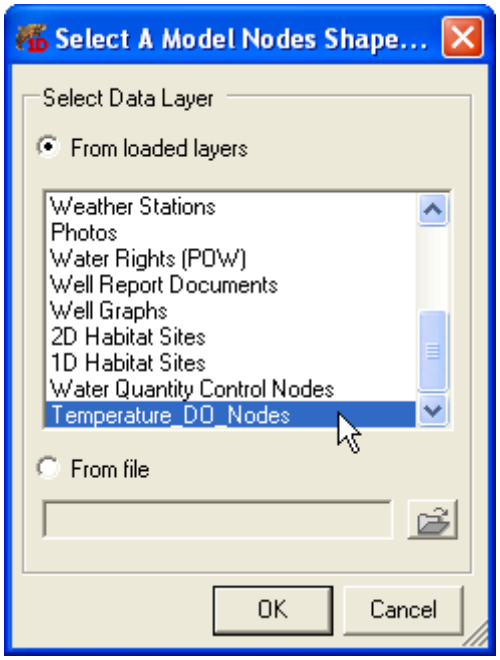
This section allows the user to select the waterbody loading output text file. Press the browse button to the right of the Watershed Loading Output Text File pathname. This will bring up the Open File form as shown below.



Once the desired file has been selected, press Open.

*Select Model Nodes Shapefile*

To change the Model Nodes Shapefile used in the Temperature and DO Flags Model, click the browse button to the right of the Model Nodes Shapefile pathname. The following form will appear, allowing the user to select a different shapefile from the loaded layers or from file.

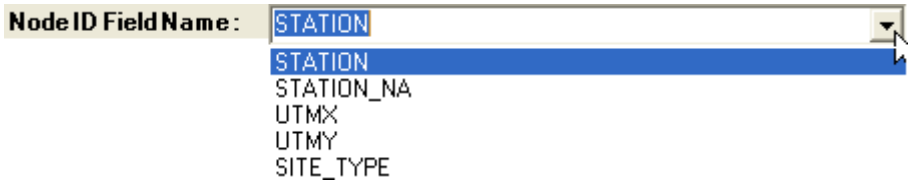



To select a Model Nodes Shapefile from the layers currently loaded in the DSS, select the From loaded layers option and select a layer from the list as shown above.

To select a Model Nodes Shapefile from file, select the From file option and enter the entire file pathname in the box below or press the browse button to navigate to the desired file location.

Once a layer or file has been selected, press OK to accept the shapefile and continue. To return to the Temperature ad DO flags form without selecting a new Model Nodes shapefile, press Cancel.

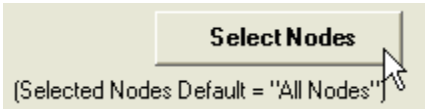
The user must specify which field in the selected Model Nodes shapefile is to be used using the Node ID FieldName box as shown below.




If more than one Field Name exists in the selected Model Nodes shapefile, it can be selected from the list by pressing the down arrow to the right of the Node ID Field Name box as shown above.

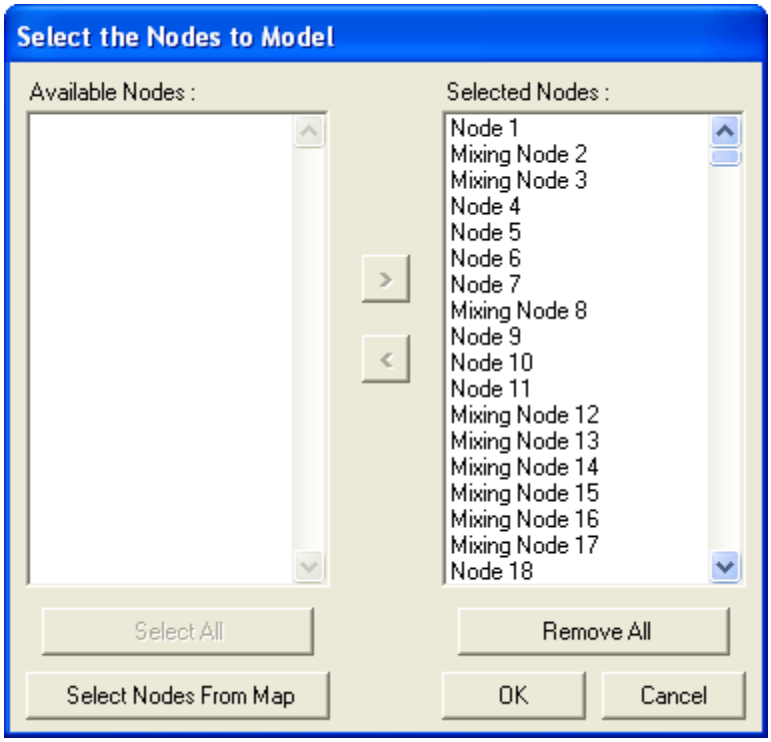
*Select Nodes*

To select the Nodes to be modeled in the Temperature and DO Flags Model, press the **Select Nodes** button as shown below.




Note that if this button is not pushed, the model will use the default of all nodes.

Pressing the **Select Nodes** button brings up the following form.



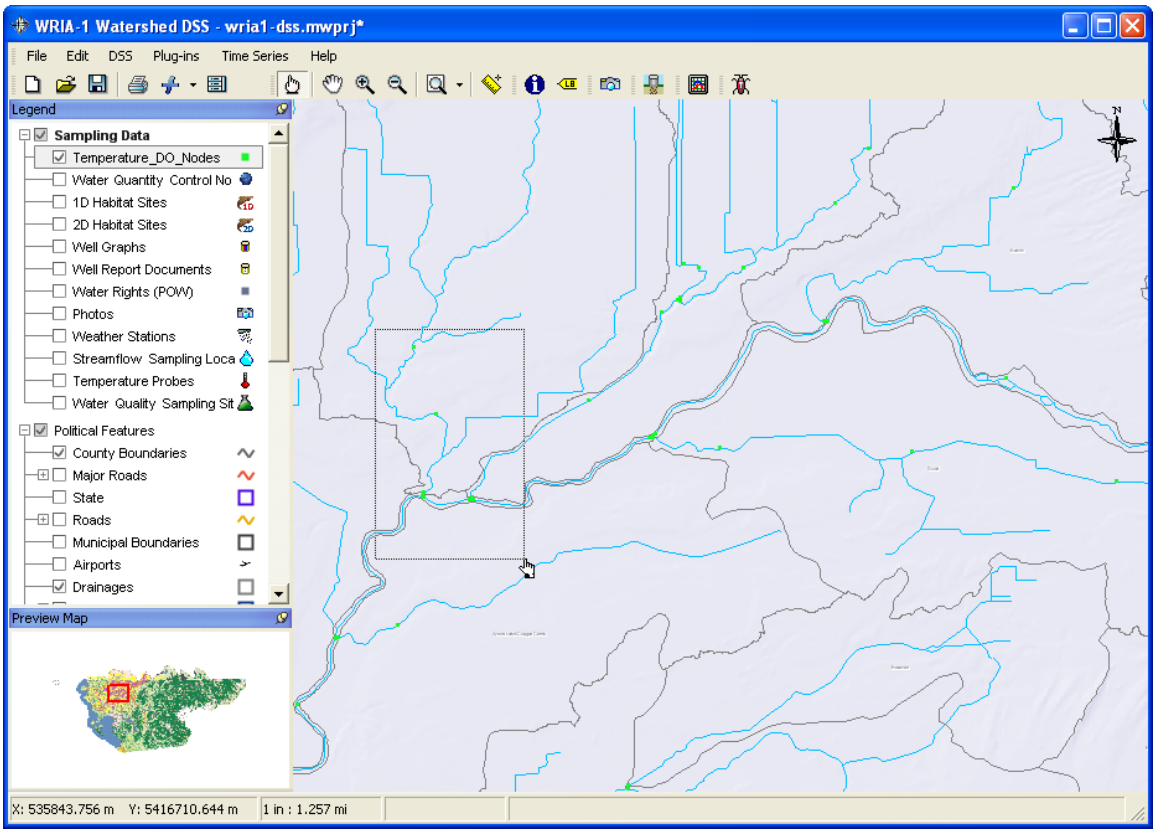

The nodes labeled using Node correspond to the water quantity nodes, one node for each drainage, and have the same numbering. Nodes labeled as Mixing Nodes are nodes where two or more drainages meet and have a combined node. Again, the default is to model all the nodes.

Since by default, all the nodes are selected, they are listed in numerical order on the right hand side of the form in the Selected Nodes window. To remove a node from the Selected Nodes window, click on the node number with the mouse to highlight that node then, press the remove button. This will remove the selected node from the Selected Nodes window. This node will now be available and will be listed in the Available Nodes window on the left hand side of the form. To remove all the nodes in the Selected Nodes window, press the button. This will remove all the nodes from the Selected Nodes window and they will be displayed in the Available Nodes window.

To select a node from the Available Nodes, click on the node number with the mouse to highlight that node, then press the add button to move the selected node to the Selected Node window.

Multiple nodes can be selected for addition or removal by holding down the Ctrl key while using the mouse to select the desired nodes.

The user may also select specific nodes from the map by pressing the Select Nodes From Map button. This will minimize all other windows allowing the user to view the map. To select a node from the map, the desired node shapefile layer must be active and selected. This layer must be the same shapefile entered in the Model Nodes Shapefile section described above. If the user attempts to select nodes from a shapefile other than the one entered in the Model Nodes Shapefile section, and error will occur. To select nodes, the user must use the Select Shapes tool. In the example shown below, nodes from the Temperature DO Nodes layer are being selected.



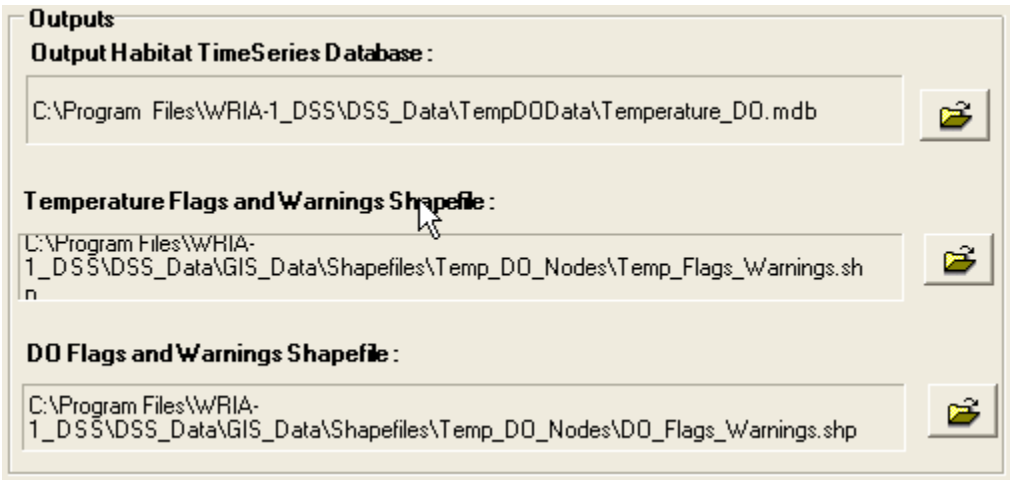

Once a node has been selected, the Select the Nodes to Model form will reappear, and the selected node will be listed in the Selection Nodes section of the form. Additional nodes can be selected by pressing the Select Nodes From Map button and clicking on another node. The user may also select multiple nodes at once by clicking and dragging the mouse to create a rectangle around the desired nodes. This will select all nodes that lie within the rectangle.

Once all the desired nodes have been selected and are listed in the Selected Nodes box, and the desired extrapolation method has been selected, press OK to accept the selections and return to the Temperature and DO Flags Model Parameters form. The user must select at least one node to model or an error will occur.

To return to the Temperature and DO Flags Model Parameters form from the Select the Nodes to Model form without selecting any nodes or an extrapolation method, press the Cancel button. The default options will then be used to model.

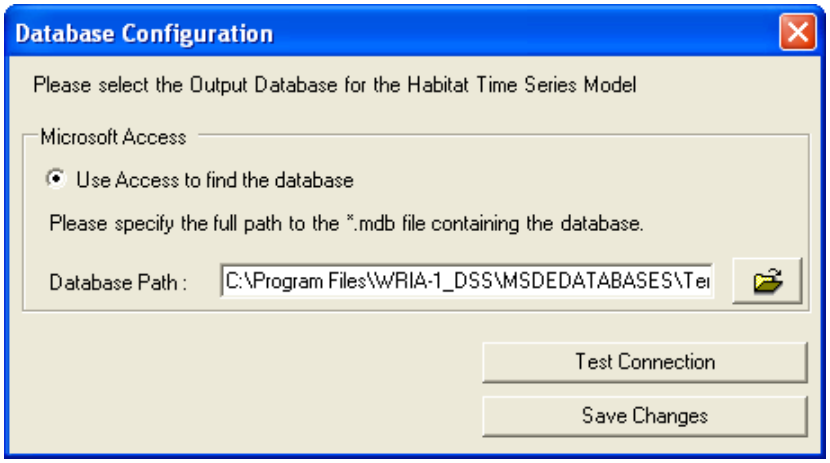
***Edit Temperature and DO Flags Model Outputs***

The Outputs section of the Temperature and DO Flags Model Parameters form, located near the bottom of the form, allows the DSS user to edit the outputs for the Temperature and DO Flags Model including selecting the Output Temperature and DO Database, Temperature Flags and Warnings Shapefile and the DO Flags andWarnings Shapefile. Each of these sections is explained in detail in the following sections.



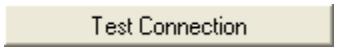

*Select Output Temperature and DO Database*

The Output Temperature and DO Database, named Temperature_DO.mdb is the database where the output data for the selected nodes will be written. Use the browse button to the right of the database name to navigate to this database. This will bring up the Database Configuration form.

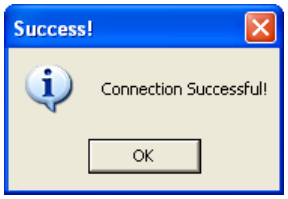

Database Configuration
Please select the Output Database for the Habitat Time Series Model
Microsoft Access
Use Access to find the database
Please specify the full path to the *.mdb file containing the database.
Database Path :
C:\Program Files\WRIA-1_DSS\MSDEDATABASES\Te
Test Connection
Save Changes

This database is used for all the Temperature DO Flags model runs. This database is an Access database and the same database, or one with identical formatting must be used. When the Temperature DO Flags model is run, any data already existing in this database will be cleared and the new data for the selected nodes will be written to the database.

The user must enter the complete pathname in the box to the right of Database Path: or use the browse button to navigate to the desired database.

To test the connection to the database and to verify that the database is available, press the 

button. If a successful connection is made to the database listed the following message will appear.



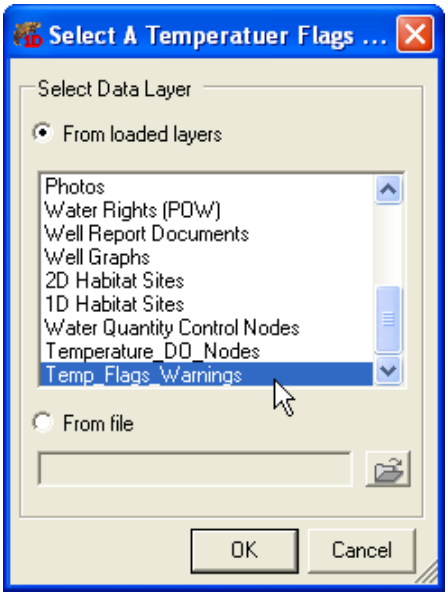

Once a successful database connection has been made to the desired database, press the

button to continue.

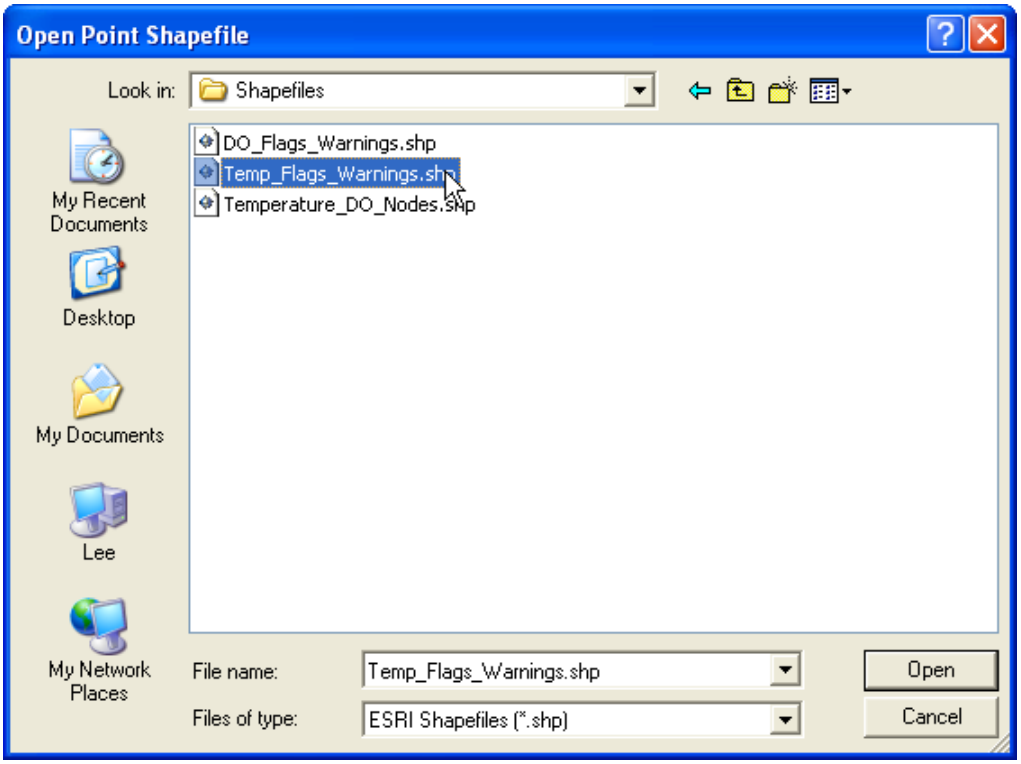
*Select Temperature Flags and Warnings Shapefile*

When the Temperature and DO Flags Model is run, temperature data, flags and warnings for the selected nodes will be displayed on the map using a shapefile. A default Temperature Flags and Warnings shapefile exists and should be used for all model runs. The data in this shapefile is overwritten each time the model runs. The shapefile is named Temp_Flags_Warnings.shp. To select this shapefile, press the browse button to the

right of the Temperature Flags and Warnings Shapefile pathname box. This will bring up the screen shown below.



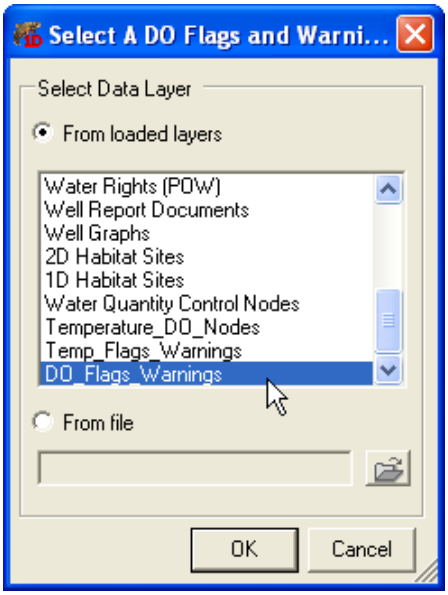

If the Temperature Flags and Warnings shapefile has already been added as a layer, it can be selected from the loaded layers list as shown above. Otherwise, click the From file radio button and enter the pathname or press the browse button to bring up the Open Point Shapefile form as shown below.



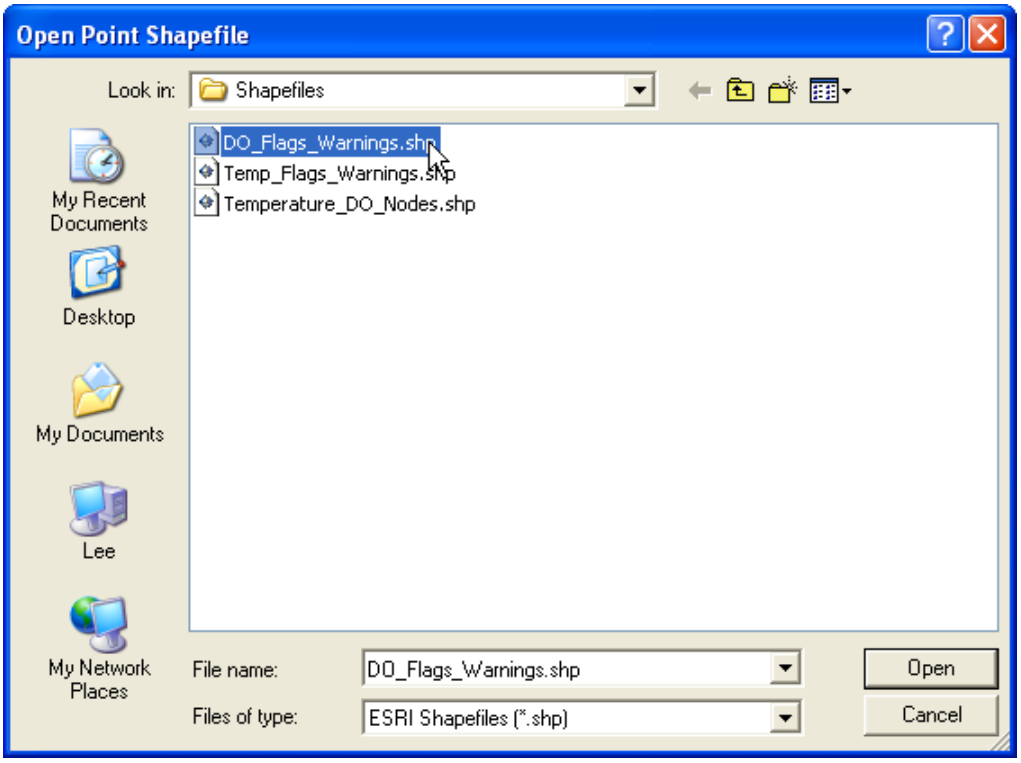
This same shapefile should be used every time the model is run since it is updated with the temperature data for the selected nodes. The shapefile used must have the same formatting as the Temp_Flags_Warnings.shp file, or the model will not run successfully.

*Select DO Flags and Warnings Shapefile*

The model also writes DO data, flags and warnings to a shapefile. This shapefile and its selection process are nearly identical to the Temperature Flags and Warnings Shapefile described above. A default DO Flags and Warnings shapefile exists and should be used for all model runs. The data in this shapefile is overwritten each time the model runs. The shapefile is named DO_Flags_Warnings.shp. To select this shapefile, press the browse button to the right of the DO Flags and Warnings Shapefile pathname box. This will bring up the screen shown below.




If the DO Flags and Warnings shapefile has already been added as a layer, it can be selected from the loaded layers list as shown above. Otherwise, click the From file radio button and enter the pathname or press the browse button to bring up the Open Point Shapefile form as shown below.

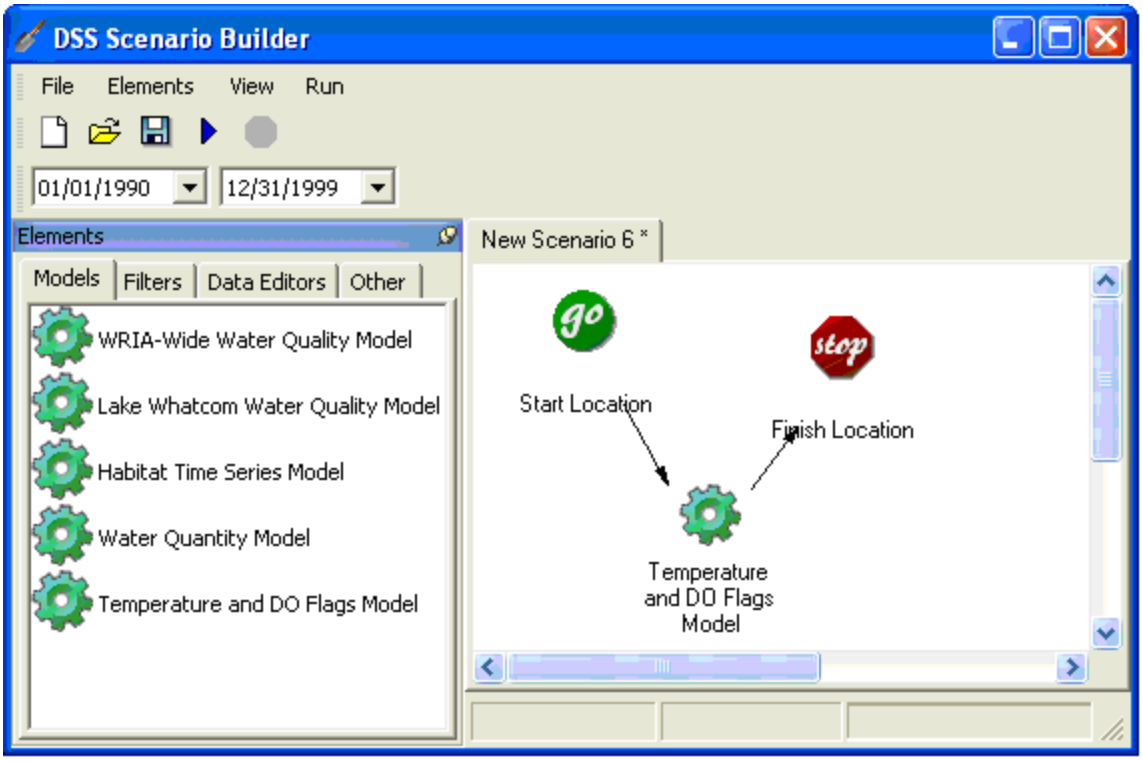



Again, this same shapefile should be used every time the model is run since it is updated with the DO data for the selected nodes. The shapefile used must have the same formatting as the DO_Flags_Warnings.shp file, or the model will not run successfully.

Once all the input and output parameters have been set using the Temperature and DO Flags Model Parameters form, press OK to accept the changes and return to the DSS Scenario Builder window. To return to this window without accepting the changes made to the Temperature and DO Flags Model Parameters form, press Cancel.

## Run Temperature and DO Flags Model

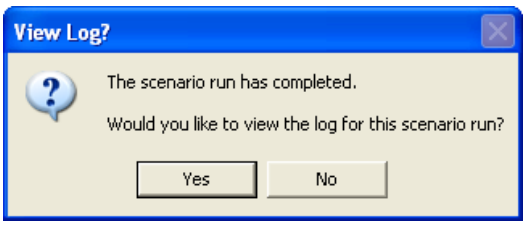
The DSS Scenario Builder allows the DSS user to run the Temperature and DO Flags Model.




Once the model inputs and outputs have been configured, the model can be run. The Temperature and DO Flags Model does not use the date range selectors, so it is not necessary to enter a date range.

To run the scenario, click the button. If you would like to abort the scenario run, click the button to cancel the scenario run.

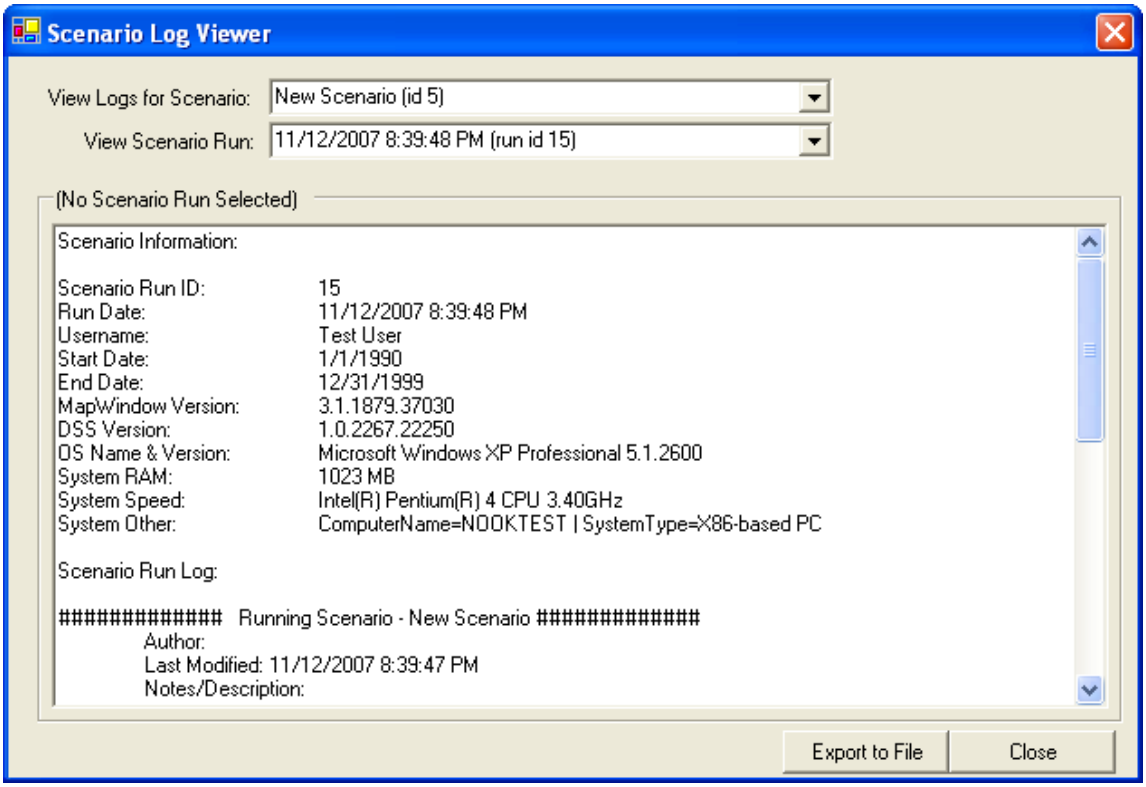
When the scenario has completed, the DSS user will be asked if they would like to view the scenario run log (a text file containing messages output by the elements contained in the scenario that has just been run).



To view the log for the scenario run, click the Yes button. To decline viewing the scenario log at this time click the No button.

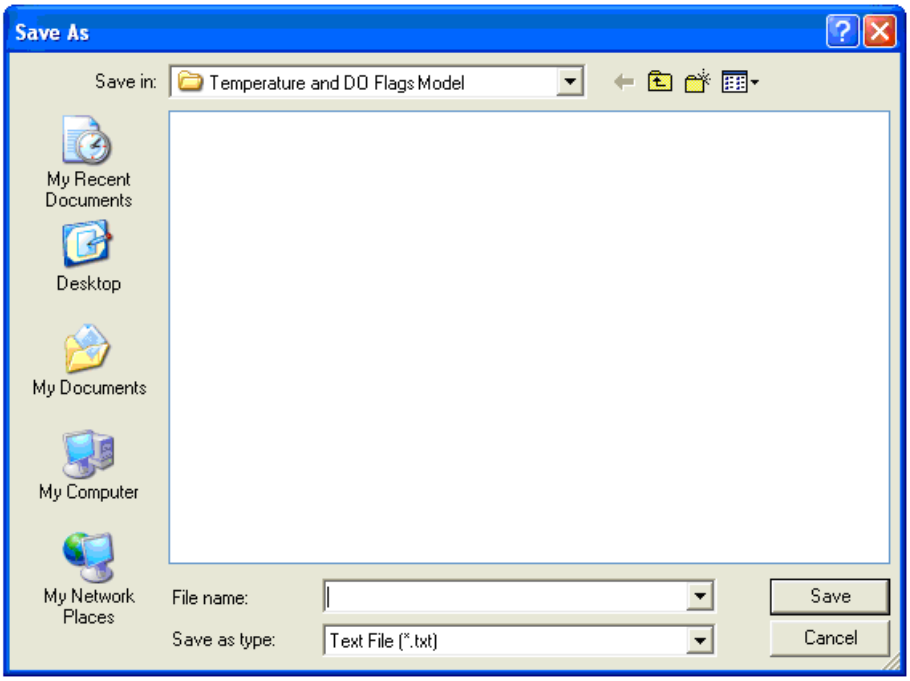
The scenario run log is saved for each individual execution of the scenario. The log contains information about the user running the scenario, their computer, and the elements involved in the scenario. Each element that runs contributes a segment to the run log, including details about input and output files and what actions the element took.

An example scenario log viewer is shown below:




### *Exporting Scenario Run Log*

The scenario run log may be exported, which is useful for archiving scenario results. Exporting the log may be done by clicking the “Export to File” button in the scenario log viewer (above). You will be asked where to save the file:



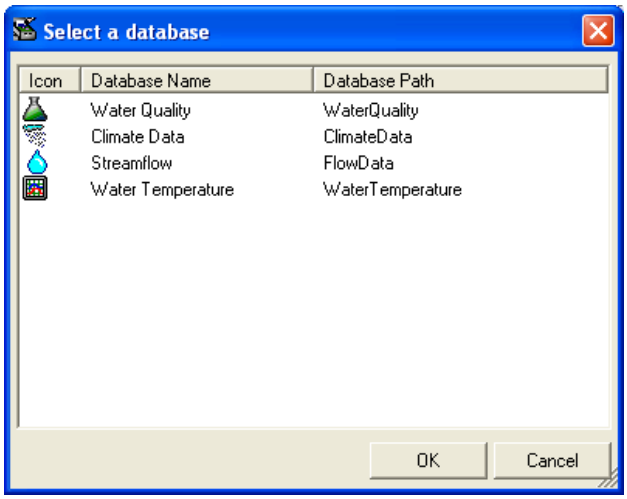

The output file will be textual (ASCII text), and will contain exactly what the scenario log viewer shows.

# Time Series Data Analyst Plug-in

The Time Series Data Analyst Plug-in to the DSS allows users to view and analyze available surface water quality data, climate data, streamflow data and water temperature data for the watershed. Databases of water quality, climate, streamflow and water temperature data are linked to a geographic coverage showing the locations at which the data have been collected. Analysis capabilities include a variety of data plotting options and statistical analyses. For the purposes of this manual, the Water Quality Database will be used to demonstrate the functions of the Time Series Analyst. The same data viewing and analysis procedures may be followed with any of the other databases listed.

## Viewing Water Quality Data

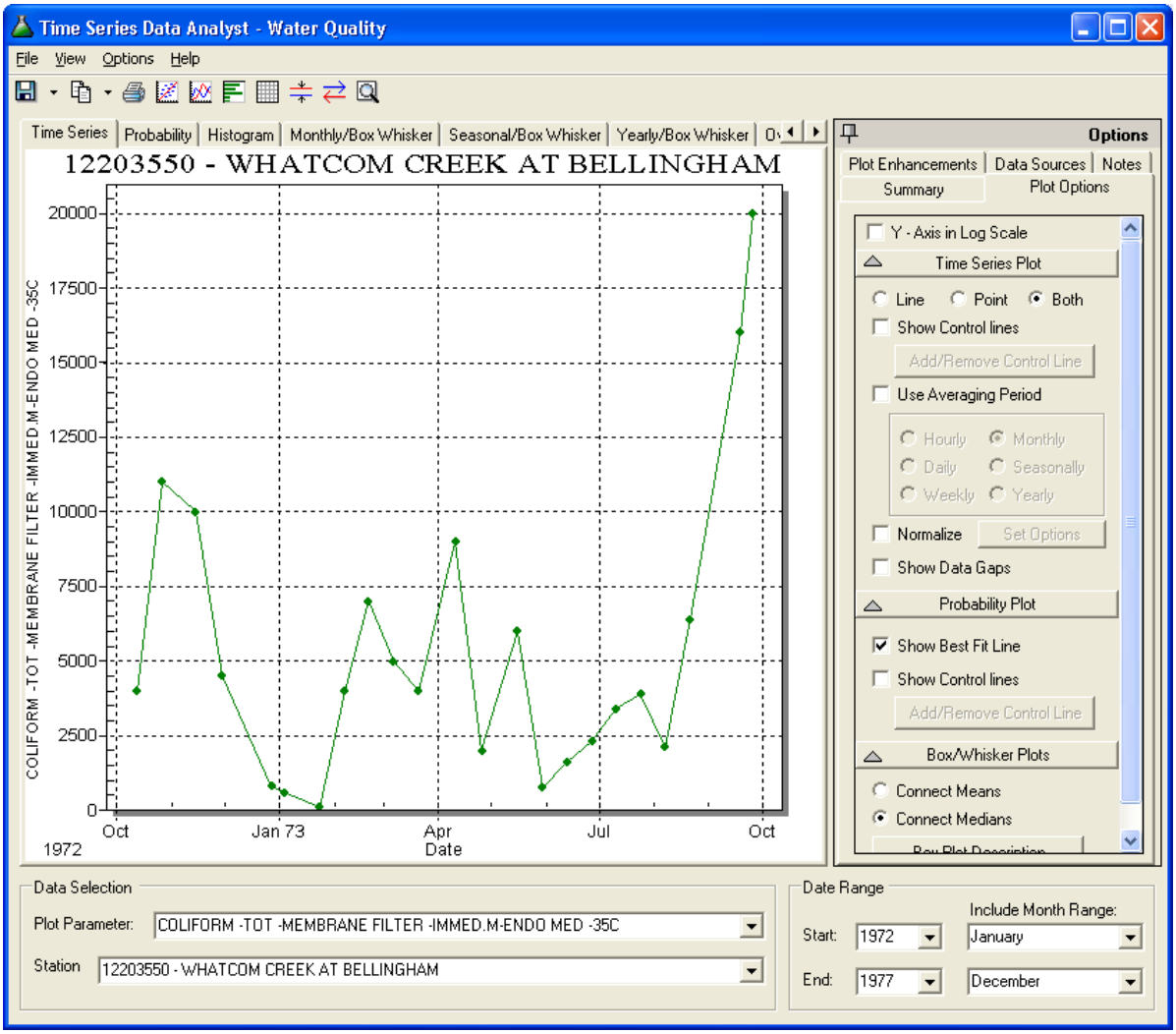
The available water quality data in the watershed can be viewed by either clicking on the Time Series Analyst toolbar button and selecting Water Quality from the list of databases (shown below) or by clicking on a water quality station from the map using the Select Shapes button.




Selecting a point from the map loads the available water quality data for the selected station into the Time Series Data Analyst - Water Quality (henceforth referred to as Time Series Analyst), whereas clicking on the Time Series Analyst toolbar button and selecting the Water Quality database loads data from all water quality stations into the Time Series Analyst.

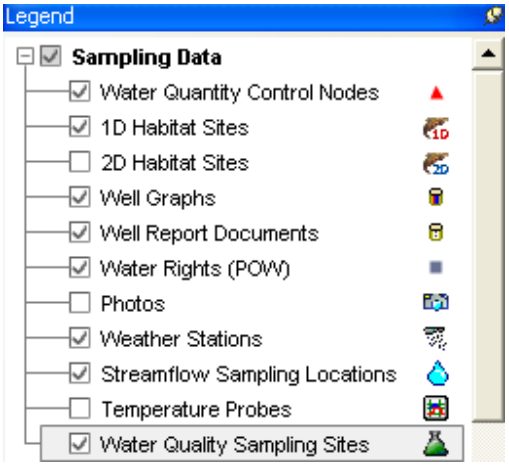
### *Viewing Data via the Time Series Data Analyst Toolbar Button*

To view and analyze all data in a database click the Time Series Data Analyst toolbar button and select a database from the list. Selecting the Water Quality database from the list opens the Time Series Analyst and makes available the entire water quality database in the form shown below. In this manner, users can browse the entire database of water quality data without interacting with the map.

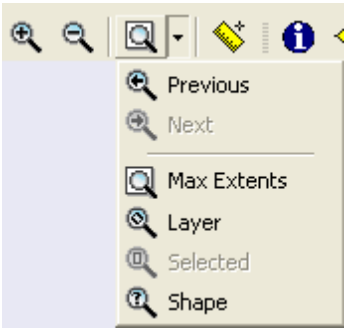



### *Selecting Water Quality Stations from the Map*

First, make sure that the Water Quality Sampling Sites layer is turned on (checked) and active (enclosed in a gray box) in the Legend frame as shown below.



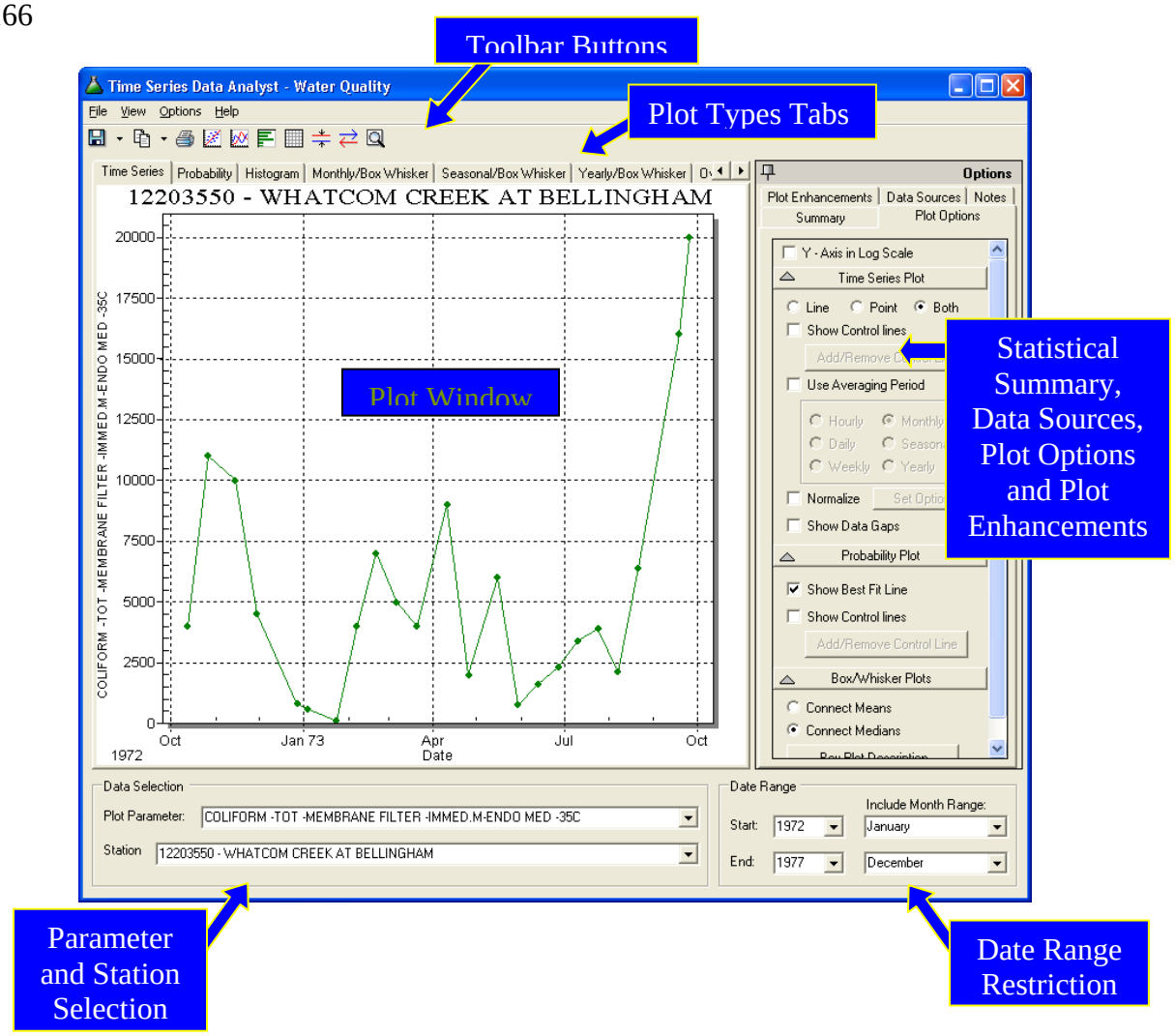

Next, use the Zoom and Pan Tools to locate the area of interest on the map. Additional Zoom tools are accessed by clicking on the black arrow as shown.




Using the select tool , points can be selected by either clicking on individual stations or by clicking and dragging a box around one or several locations. Selecting points from the Water Quality Sampling Sites theme will load the available water quality data for those selected locations into the Time Series Analyst form shown above.

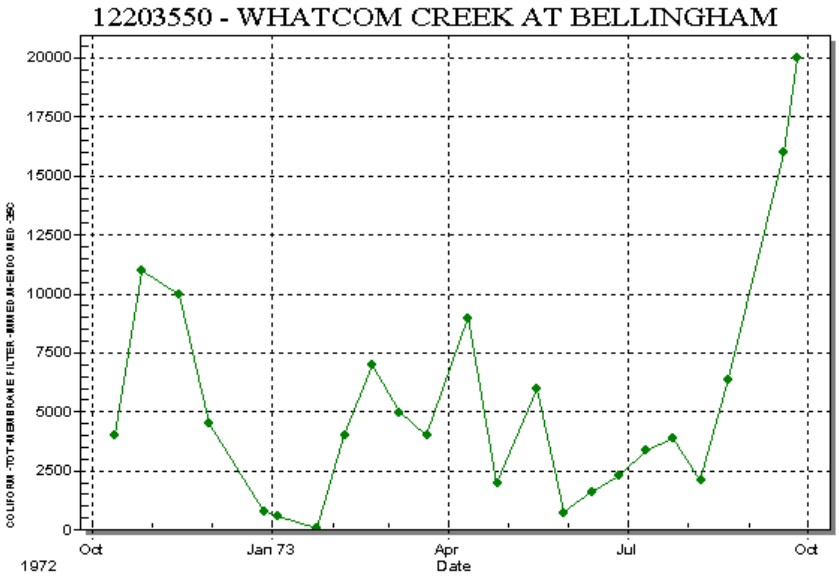
## Time Series Analyst Options

Many options are available for viewing and analyzing data using the Time Series Analyst. The following figure shows the major parts of the working screen and some of the options available in the Time Series Analyst. Each of these is described in the following sections.



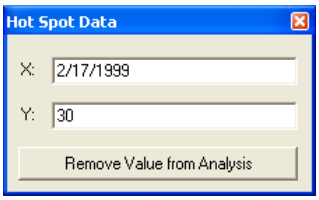

### *Plot Window*

The plot window, shown in the figure below, displays a plot of the data for the currently selected station, water quality parameter, and plot type.




The plot shown in this window can be customized using options discussed in following sections. A user can zoom in on any area in any of the plots by simply clicking and dragging a rectangle to encompass the area of interest on the plot area. To undo the zoom, click on the Undo Zoom toolbar button .

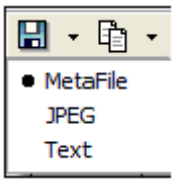
In addition, users may identify any point shown in the plots by simply clicking on the data point. When the mouse is over the data point, it will change from an arrow to a hand. Clicking the mouse on a data point will bring up the X and Y values for that point in a Hot Spot Data window as shown below.



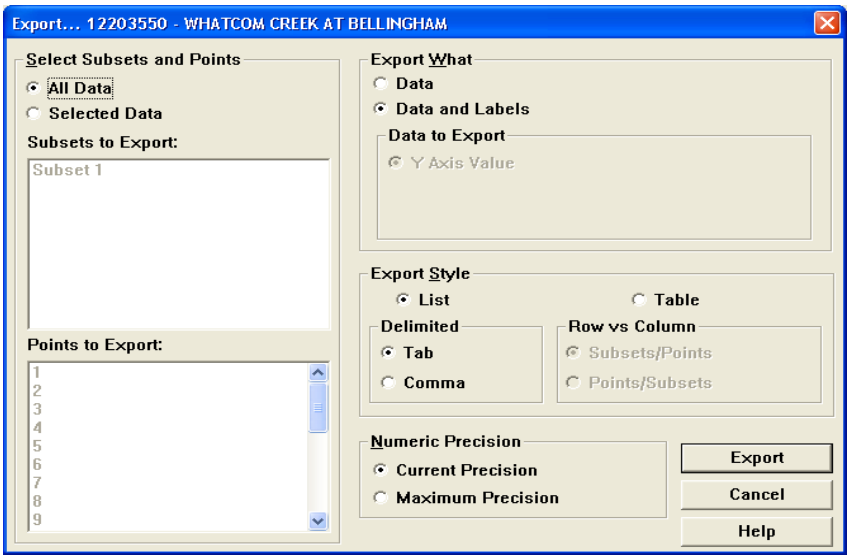

By clicking on the **Remove Value from Analysis** button, users may remove the selected data point from the plot.

### *Plot Saving, Exporting, and Printing*

The save button on the tool bar allows the user to export the current plot to a jpeg file or a metafile. The small down arrow next to the save button allows the user to specify which file type to be used in exporting the current plot.



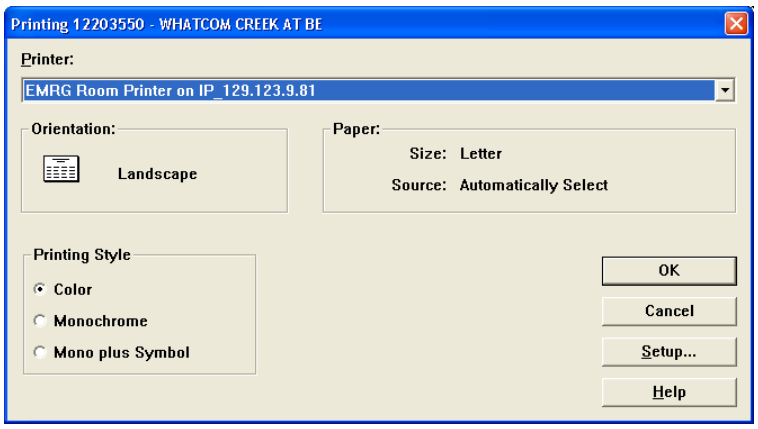

In addition to exporting the plot as an image, the user may select **Text**, in which case a text file containing the data used to generate the plot can be created. Selecting **Text** will allow the user to specify a location and filename for the data and will then open a dialog in which the user can specify export options for the text file as shown below.




To export the file with the specified options, press the **Export** button. To discard the file and return to the previous plot window, press **Cancel**. For help with these options, press **Help**.

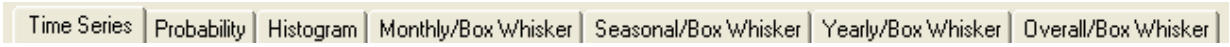
The copy button on the toolbar allows the user to copy the current plot to the clipboard. Once copied, plots can be pasted into documents for reporting and printing purposes. By clicking on the small down arrow next to the copy button the user can specify the format of the copied plot (jpeg, metafile, or text). Similar to the save button, if the user selects **Text** using the down arrow the **export text** dialog shown above is opened. However, when the Export button is clicked the data are copied to the clipboard rather than saved to a text file.

The print button on the toolbar opens a dialog so that you can specify a printer to which the current plot will be sent for printing and the options available for that printer. This dialog is shown below.




### *Plot Types Tabs*

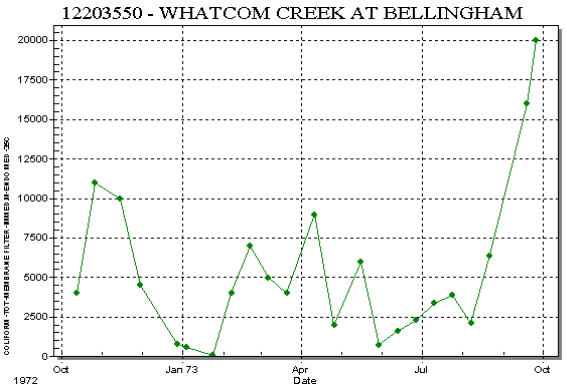
The Plot Types tabs allow the user to view several different types of plots for the water quality data. The different plot types are viewed by clicking on the tab for the desired plot type at the top of the plot window.



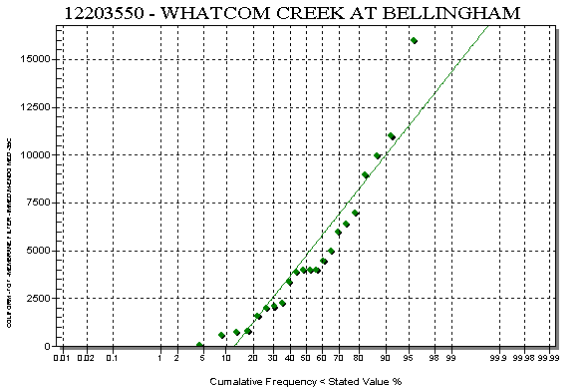

Plot types currently available via the plot types tabs in the Time Series Analyst include:

1. Time series
2. Probability
3. Histogram
4. Monthly Box/Whisker
5. Seasonal Box/Whisker
6. Yearly Box/Whisker
7. Overall Box/Whisker

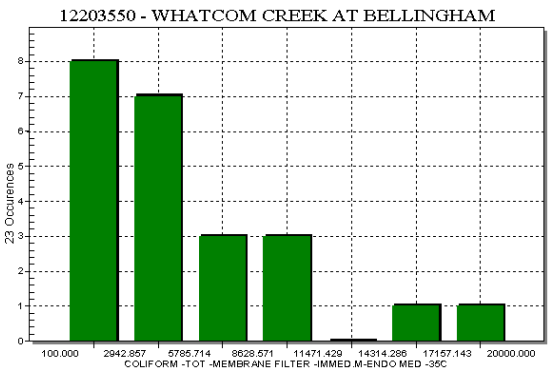
The following figures show examples of each of these plot types.



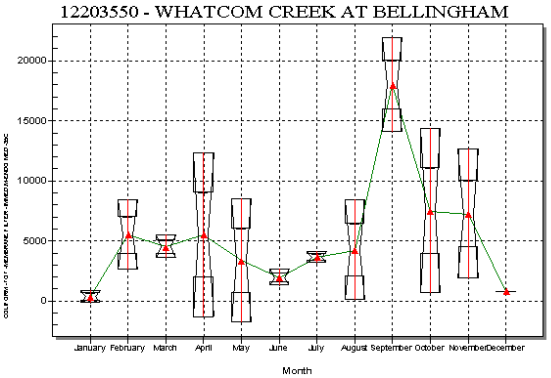

(1) Time Series



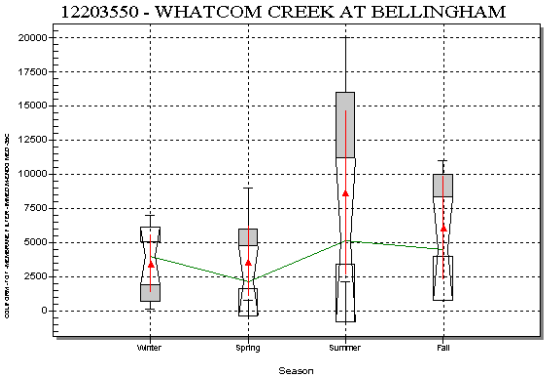

(2) Probability



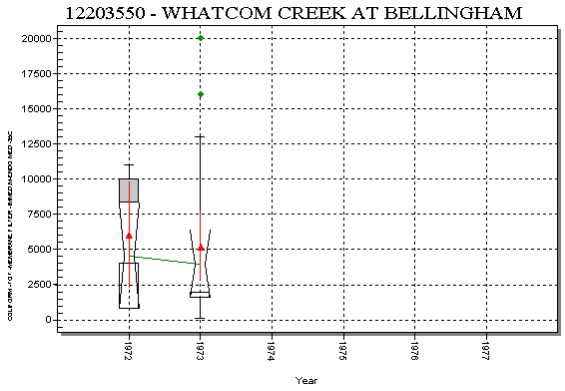

(3) Histogram



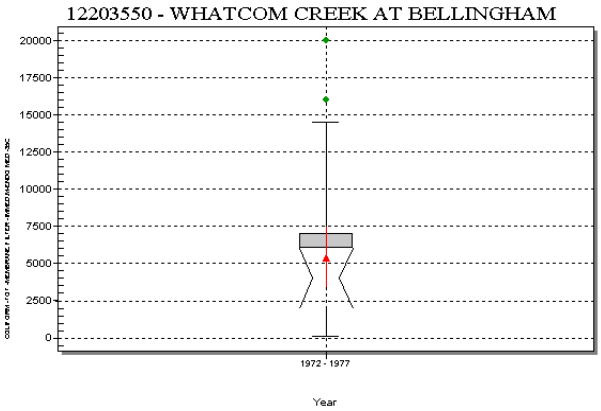

(4) Monthly Box/Whisker




(5) Seasonal Box/Whisker



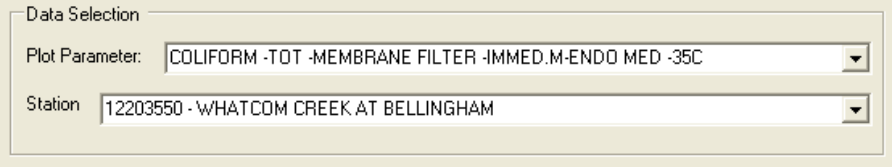

(6) Annual Box/Whisker




(7) Overall Box/Whisker

### *Selecting Stations and Water Quality Parameters*

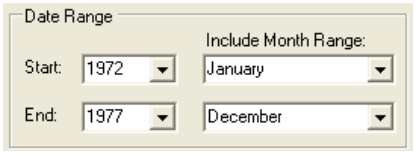
Users can select from the list of available water quality stations by using the Station pull down box located in the Data Selection box at the bottom of the window. The Station pull down box will have a list of all stations selected from the map or, if the Time Series Analyst was launched from the toolbar, the Station pull down box will have a list of all stations in the water quality database. To change from one station to the next, users can simply click on a new location in the Station pull down box.




Once a station is selected, the available water quality data at that station may be viewed by selecting a water quality parameter from the Plot Parameter pull down box. This list is automatically populated with the list of parameters for which data are available at the selected station.

### *Restrict Data Range*

The Date Range pull down boxes allow you to restrict the plots and calculated statistics to a specified data window for analysis such as a single year out of a group of years.




When a station is selected, the date range boxes are automatically populated to correspond with the time period of available data for that station. Selecting a subset of the available data is accomplished by selecting a starting year and month and an ending year and month within the range of available data for the selected water quality station. Changing the date range restricts the data shown in the plots and in the statistical summaries described below.

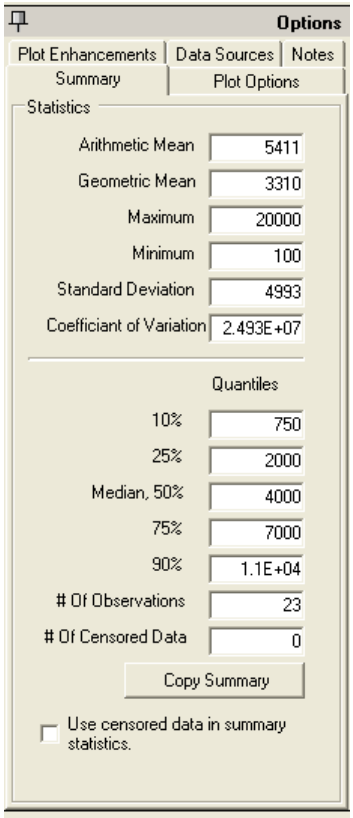
### *Statistical Summary, Data Sources, Plot Options, Plot Enhancements and Notes*

These five tabs in the Options frame to the right of the plot window provide access to several different options. Information in each of these tabs is accessed by clicking on the desired tab at the top of the control. Each tab is described below.

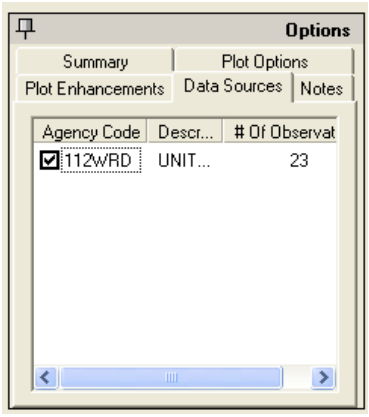
The **Summary** tab (shown below) provides descriptive statistics for the data shown in the plot window. These descriptive statistics are automatically updated each time a new

station is selected, when a new date range is selected using the data range restrictors, when a data point is removed from the plot, or when an agency is turned on or off using the Data Sources tab. The Copy Summary button at the bottom of this tab allows the user to copy the statistical summary to the clipboard so that it can be pasted into a table or document.

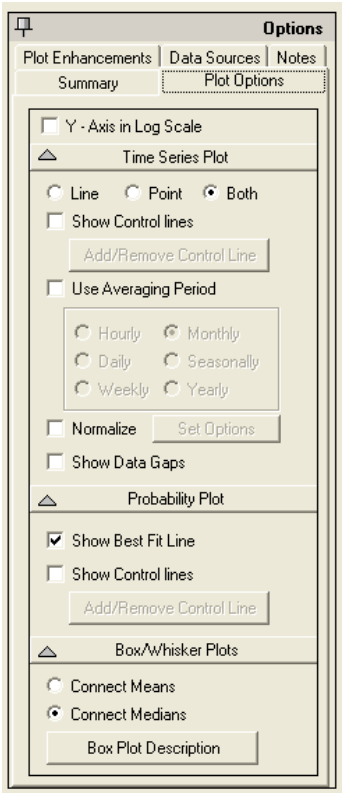
An additional option at the bottom of this form allows you to consider any censored data in the calculation of the descriptive statistics. Clicking on the check box turns this option on, and the statistics are re-calculated taking into account any below detection limit values in the database.



The **Data Sources** tab (shown below) allows users to turn on or off data collected by different agencies. This is useful in situations where multiple agencies collect data at a single location. By selecting the check boxes next to each agency, the user can control which data is shown in the plot window. If a particular agency is turned off, the data for that agency are no longer shown in the plot window and are no longer included in the statistical summary in the Summary tab. In this manner, a user can examine data collected by a single agency or compare data collected by different agencies at the same location. This tab also provides the number of observations made by each agency. The width of the data fields may be altered by clicking on the line between two field labels and dragging the mouse until the field is the desired width.



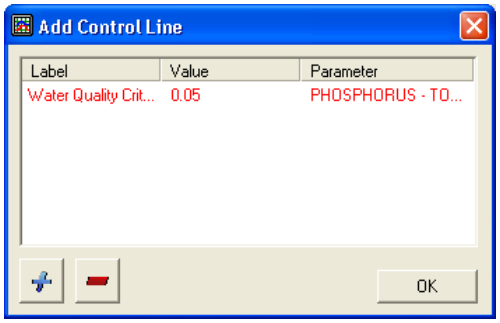

The **Plot Options** tab (shown below) allows users to modify the appearance of the plot in the plot window. Users can easily switch between log and linear scale for the Y-axis by using the check box at the top of the tab. The other options are for specific plot types. These options include control over how the time series plot is drawn (using a line, points, or both); adding user control lines (i.e., showing water quality numeric criteria on plots); whether the probability plot shows the best fit line or user control lines; and whether the mean or median values are connected in the box/whisker plots. The **Box Plot Description** button provides an explanation of the various parts of the box/whisker plots. The sections of the Plot Options tab can be expanded or collapsed by clicking on the arrow next to the name of the section (i.e., Time Series Plot, Probability Plot, Box/Whisker Plot).



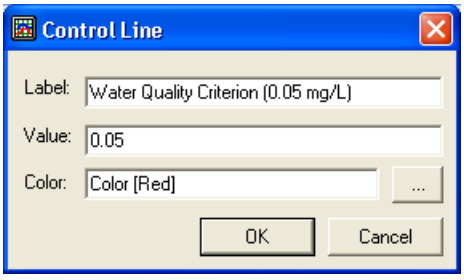

The Plot Options tab allows the user to display information in the plot window by averaging over different time periods. This is particularly useful when viewing the high frequency temperature data in the temperature probe database. Users can plot the raw data or use any of the averaging periods available to plot the averages rather than the raw data. To use this function, check the box next to "Use Averaging Period" then select the time period over which the data will be averaged.

If the "Show Control Lines" option is checked for either the Time Series or Probability plot, the user can add control lines to the plot.

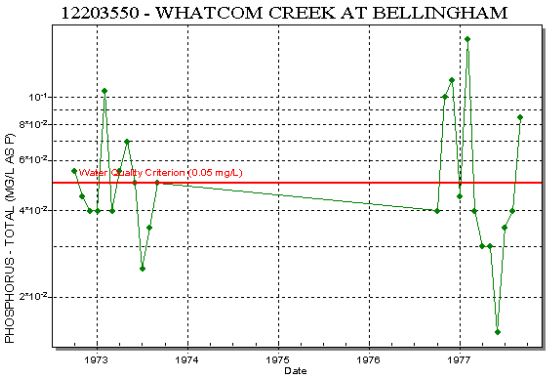
Clicking the Add/Remove Control Line button in the Plot Options tab allows the user to define any number of control lines to include in the Time Series or Probability plots using the Add Control Line form shown below.

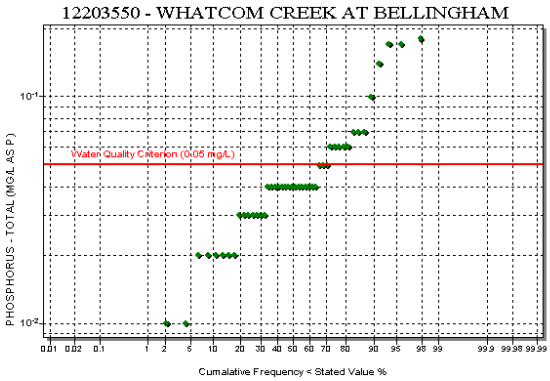



Click the button to add a new control line to the list. Fill in the Label, Value, and Color fields in the Control Line form shown below. The button to the right of the Color field allows users to select a color for the control line from a variety of predefined colors.



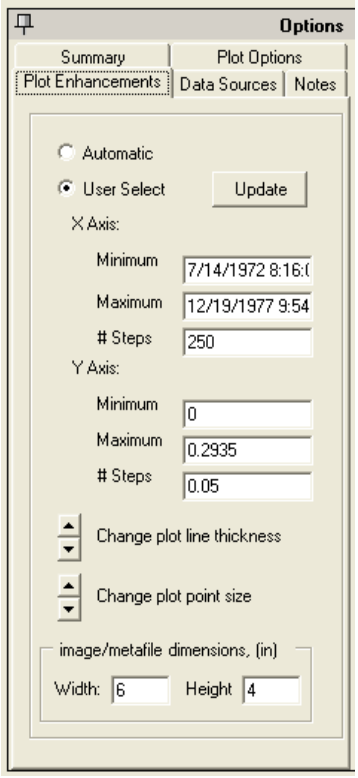

The following plots show examples of time series and probability plots showing total phosphorus data along with a control line indicating a hypothetical instream water quality numeric criterion for total phosphorus. The control lines can be removed from the plots either by clicking the Show Control Lines option (to uncheck the box) on the Plot Options tab or by clicking on the Add/Remove Control Line button, selecting the control line you wish to delete in the Add Control Line Form and then clicking the button to delete it.



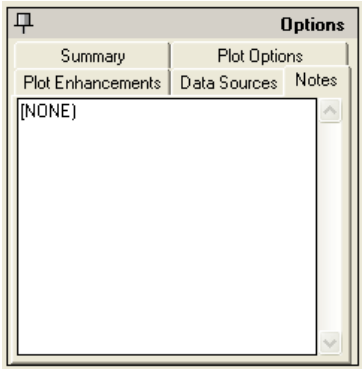


12203550 - WHATCOM CREEK AT BELLINGHAM
PHOSPHORUS - TOTAL (MG/L AS P)
Water Quality Criterion (0.05 mg/L)
Cumulative Frequency < Stated Value %

The **Plot Enhancements** tab shown below provides users with further ability to customize the plot in the plot window. On this tab, the user can control the scale of the X and Y axes by clicking on the User Select option and changing the associated values in the X Axis and Y Axis boxes. Click Update to implement the changes in the plot window. In addition, the user has control over the thickness of the plot line and the size

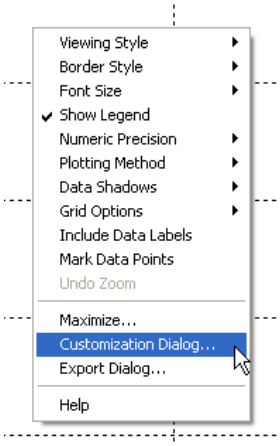
of the points shown on the plot. Click on the up or down arrow to increase or decrease plot line thickness or plot point size as desired. Changes to these aspects appear in the plot window immediately. The last option provides the user with control over the dimensions of the exported plots. These dimensions can be set to the desired image size for pasting into a document to avoid needing to resize them after they are pasted into the document.



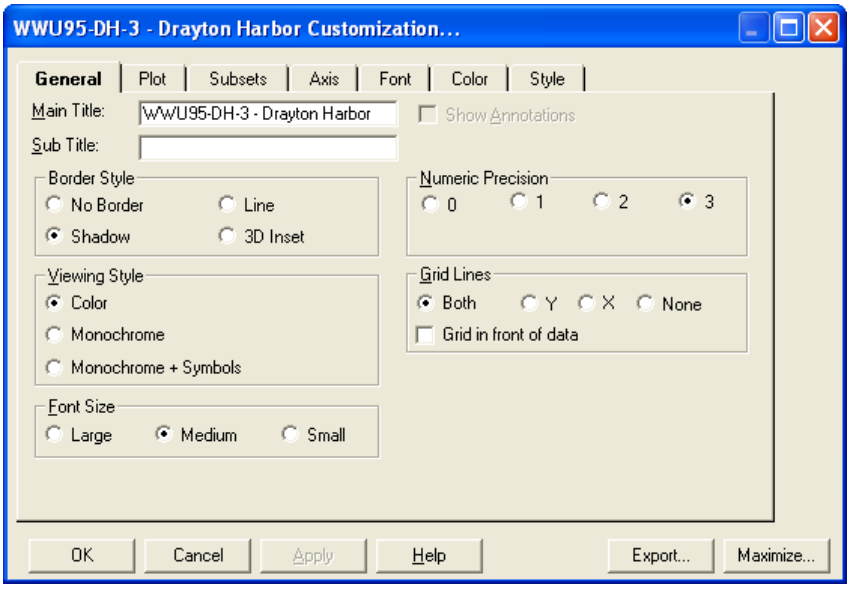

The **Notes** tab opens a screen into which a user may type any notes pertaining to analysis. The screen is populated with "(NONE)" until a user supplies notes. If notes extend beyond the original screen length, a vertical scroll bar will appear along the right side of the window so the entire length of notes can be accessed.



Additional plot customization options can be accessed by positioning the pointer on the plot window, clicking the right mouse button and selecting Customization Dialog as shown below.




This brings up the Customization Dialog window as shown below. In general, this window has several tabs that allow further manipulation and customization of the plot window. Note that these tabs may vary depending on what type of plot is being customized. Not all options are available for all plot types, and in some cases, the order of tabs will vary. Again, the following is a general description of most of the plot options available through the Customization Dialog window.



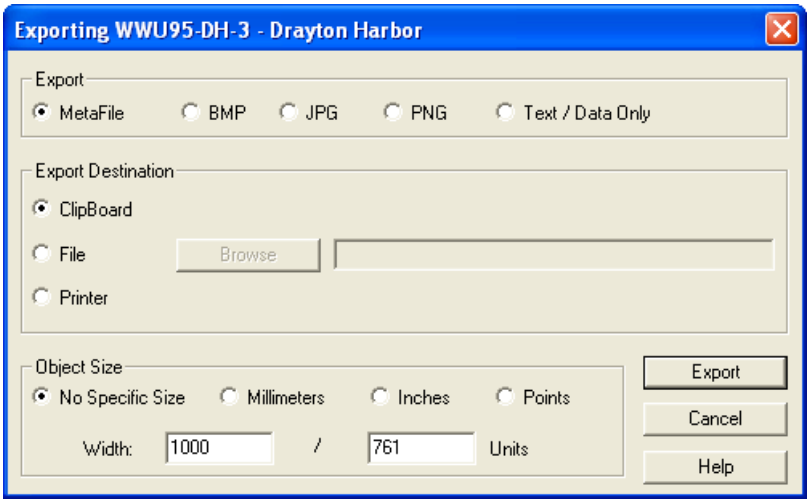

Regardless of which of the tabs are selected, the buttons along the bottom of the window perform the same functions.

Pressing **OK** will accept the selected options and return the user to the plot window.
Pressing **Cancel** discards the selected options and returns the user to the plot window.

The **Apply** button allows the user to preview the selected options without accepting them. Pressing **Help** will bring up a help window allowing the user to find more information on these options.

The **Maximize** button will resize the plot window to occupy the entire screen. To return the plot to the previous size, press the Esc button or click on the title bar at the top of the plot window.

Pressing the **Export** button allows the user to export the plot using the options shown below.



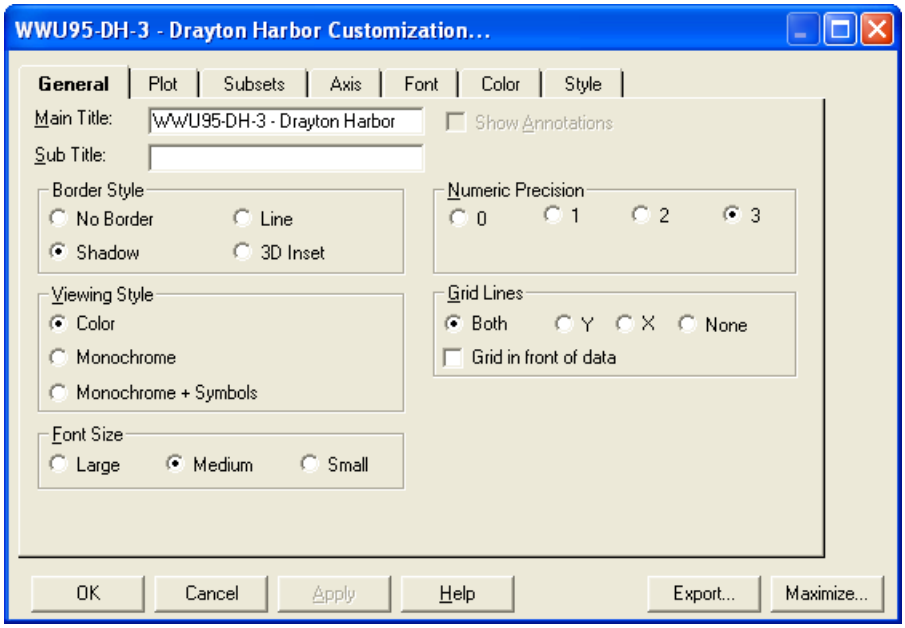

The user can choose the file type for the exported plot using the top set of radio buttons. These file types include MetaFile, BMP, JPG, PNG and Text/Data Only options.

Using the next set of radio buttons, users can select the destination for the exported plot. If the ClipBoard option is selected, the plot will be copied to the clipboard and can then be pasted into documents or reports. If the File option is selected, the **Browse** button will be activated allowing the user to navigate to the desired location and give the file a name.

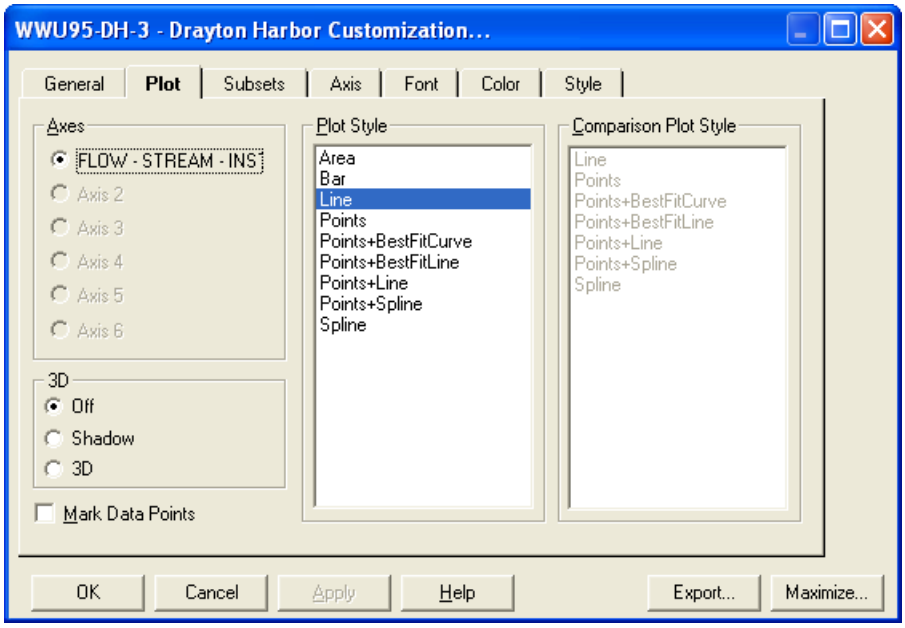
Both the ClipBoard and File options allow the user to also choose the Object Size. This can be done by choosing an option from the lower portion of the screen. The first sizing option is No Specific Size, which will leave the plot the size it appears in the window. The next sizing option is to enter the desired width and then the desired height of the plot in the text boxes to the left of Width: The user must also select the desired unit corresponding to the width and height values entered. Once the desired sizing options have been entered, the **Export** button will execute the selected options. Pressing **Cancel** will discard the selected options and return the user to the plot window. Pressing **Help** will activate the Help dialog allowing the user to find more information about these options.

If the Printer option is selected, the Object Size options change allowing the user to designate the plot size to be a Full Page or again to enter the desired width and height and select the corresponding units. Once these options have been set, press the **Print** button to select a printer and set the printing options for the selected printer.

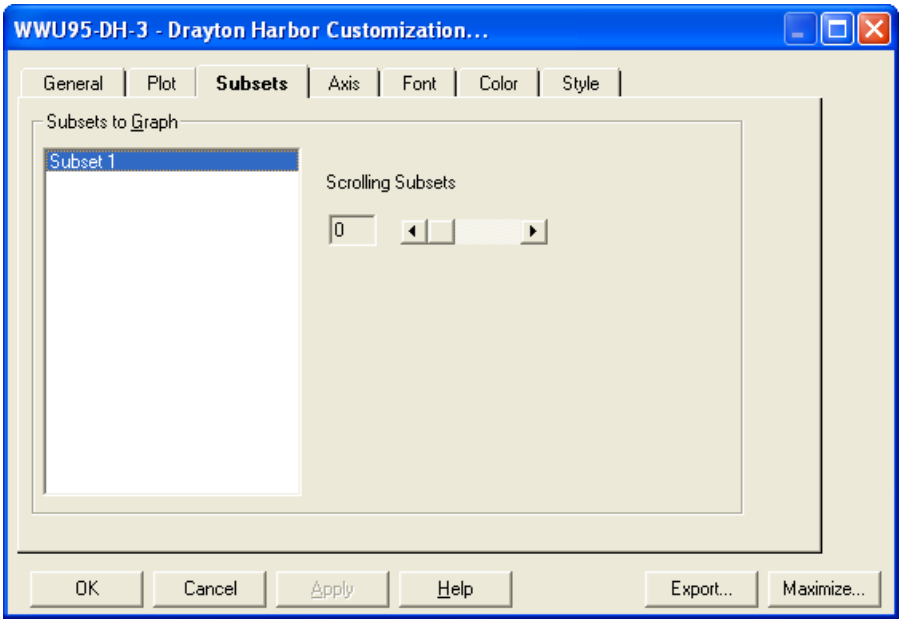
The tab titled **General** allows the user to edit the plots title, add a sub title, change border styles, viewing styles, font sizes, numeric precision used in the axes labels, and grid line options.



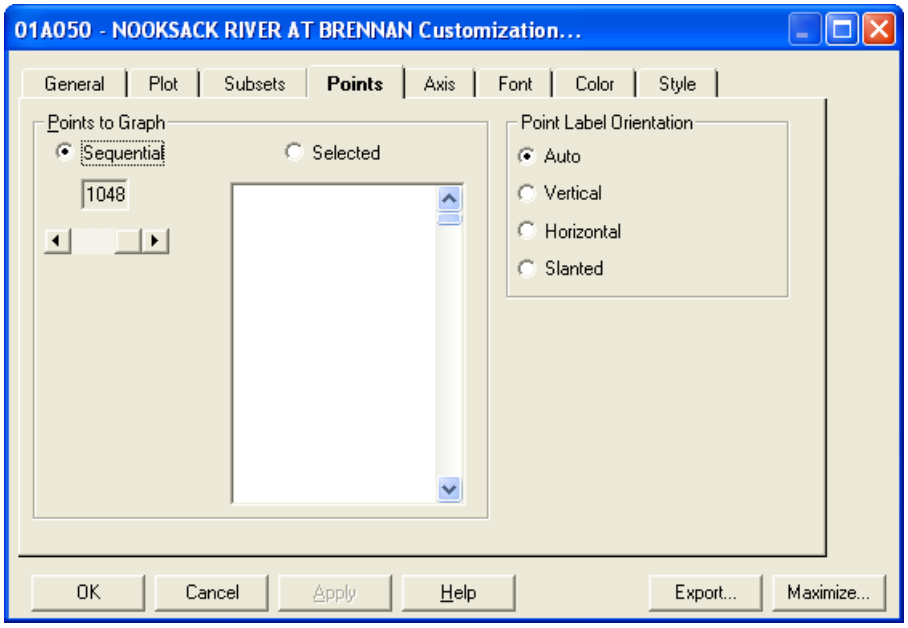

The tab titled **Plot** allows the user to change the plot style, comparison plot style, axes options, 3D display options, and have the data points marked. Not all of these options will be available for every plot generated.



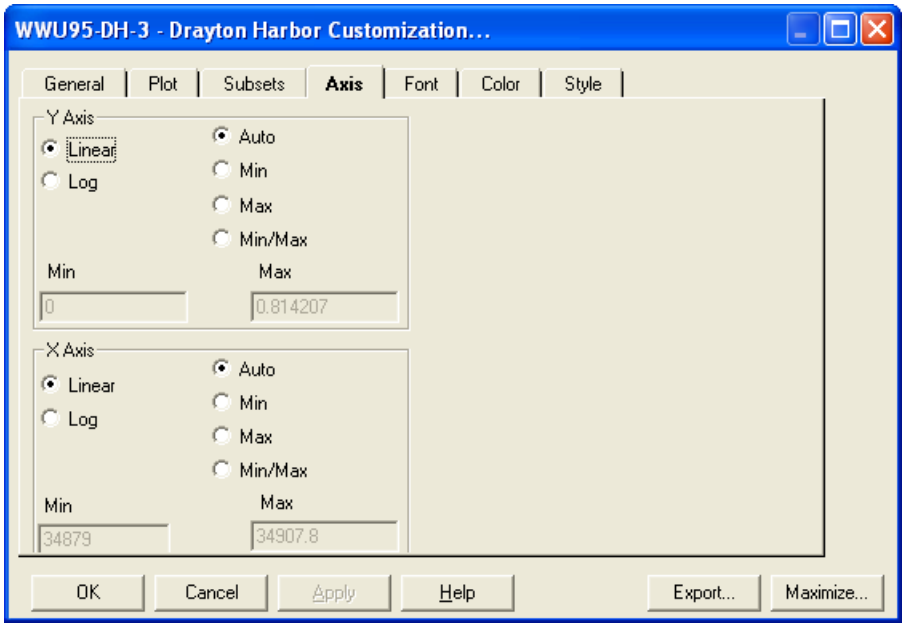

The tab titled **Subsets** allows the user to select which subsets to graph if more than one subset is present. If the graph only has one subset, this tab will not allow anything else to be selected.



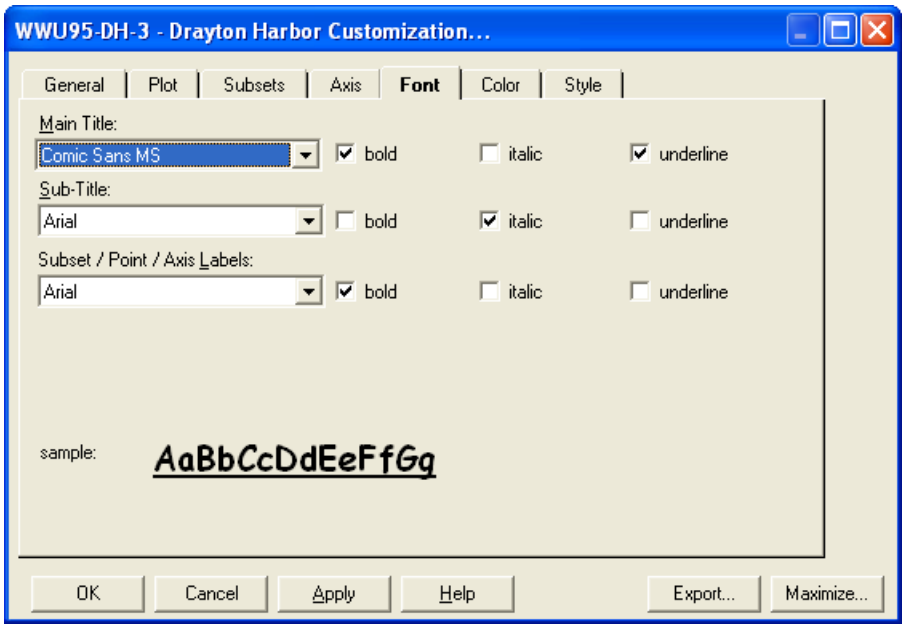

The tab titled **Points** allows the user to choose which data points to display as well as allows the user to choose how the point labels are oriented using automatic, vertical, horizontal or slanted options.



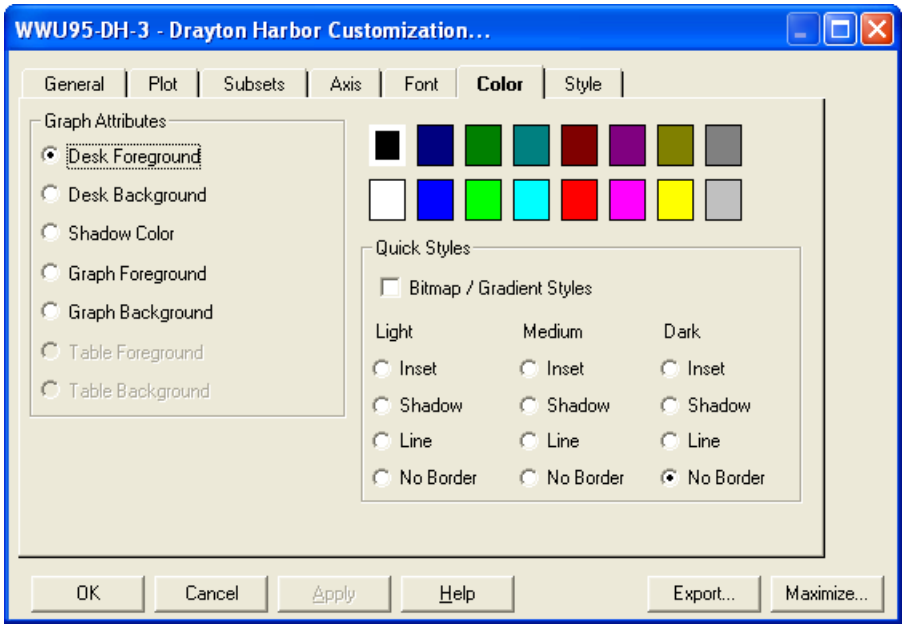

The tab titled **Axis** allows the user to change to Y and X axes from linear to log scale. This tab also allows the user to choose the range of data displayed on each axis. Auto uses the default data range, Min allows the user to input the desired minimum value to plot, Max allows the user to input the desired maximum value to plot, and Min/Max allows the user to input both a minimum and maximum value to plot.



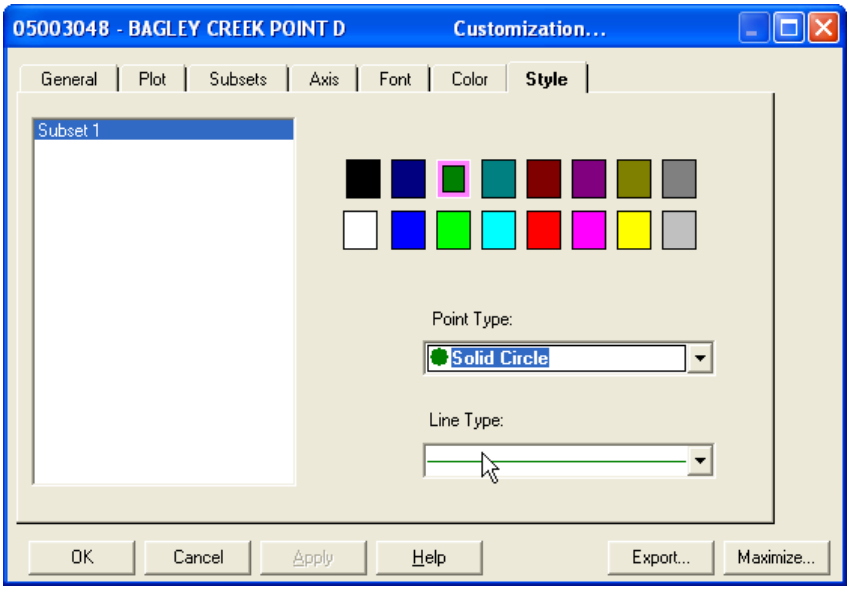

The tab titled **Font** allows the user to select a font type for the main title, sub title and subset/point/axis labels. This tab also allows the user to bold, italic, and/or underline any of these fonts. At the bottom of the tab a sample of the selected text style will be displayed.




The tab titled **Color** allows the user to change the colors of the plot. Notice the **Apply** button is not active for this tab as all color changes are automatically applied to the plot.

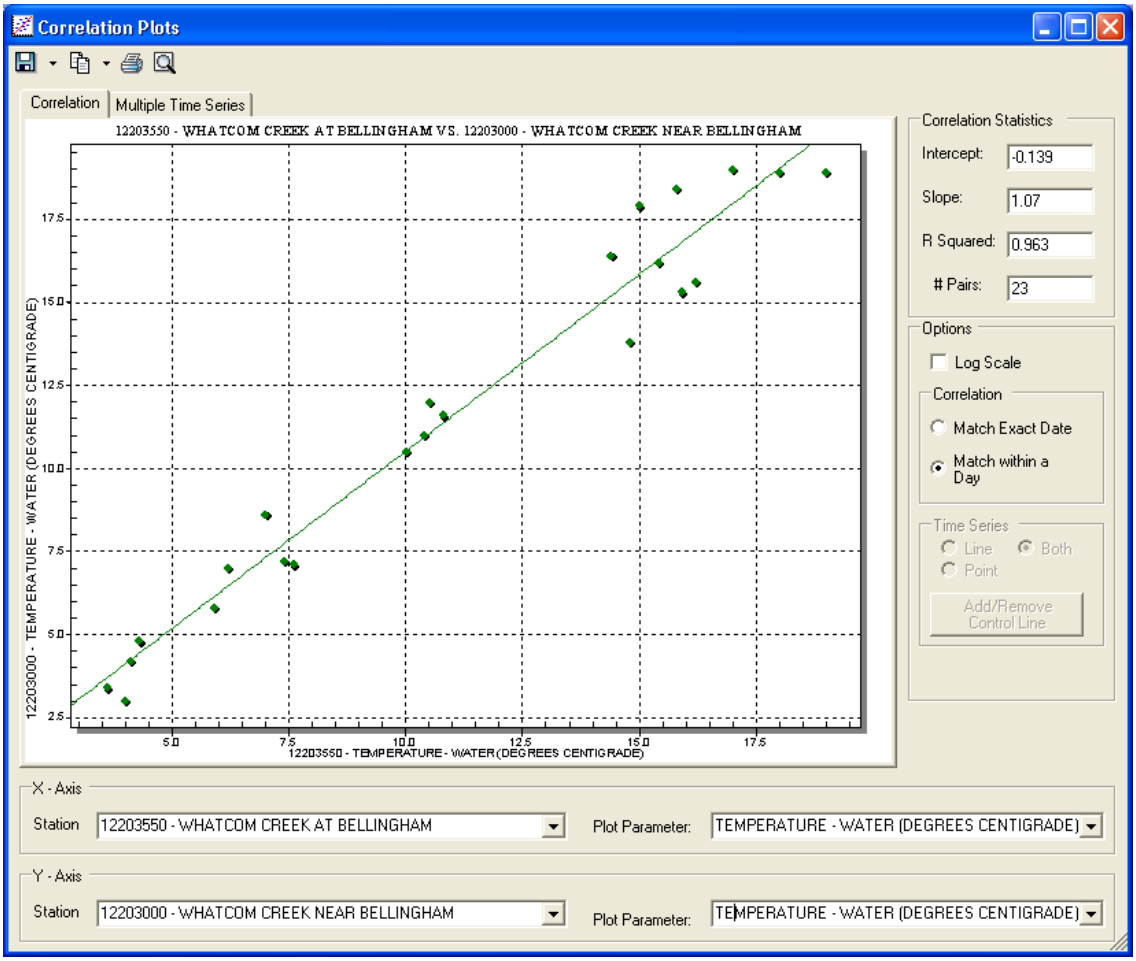



The tab titled **Style** allows the user to modify the plot styles used for each subset by selecting the point type, line type and colors.




## Correlation Analysis

The Correlation Analysis tool allows users to perform correlation analysis and multiple time series analysis on water quality data. This tool is accessed by clicking on the Correlation Analysis toolbar button . The Correlation Analysis form is shown below.

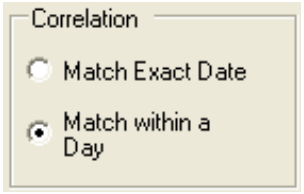



### *Correlation Analysis*

Stations used in the analysis are selected using the pull down boxes labeled X-axis and Y-axis at the bottom left of the form. In addition, users must select a water quality parameter for each selected station to be used in the correlation analysis. In this manner, users can correlate the same parameter at different stations, different parameters at the same station, or different parameters at different stations depending on their selections and provided that paired water quality observations (observations taken on the same day and time) exist.

A scatter plot of paired data for the selected water quality parameters from the selected stations is produced, with the station and parameter selected in the Y-axis pull down box plotted on the Y-axis and the station and parameter selected in the X-axis pull down box plotted on the X-axis. A linear best-fit line is generated and drawn on the plot. Descriptive statistics of the regression are shown at the top right of the form and include the slope, intercept, $R^2$, and the number of data pairs that are included in the analysis. In addition, there is an option for viewing the correlation plot on a log scale.

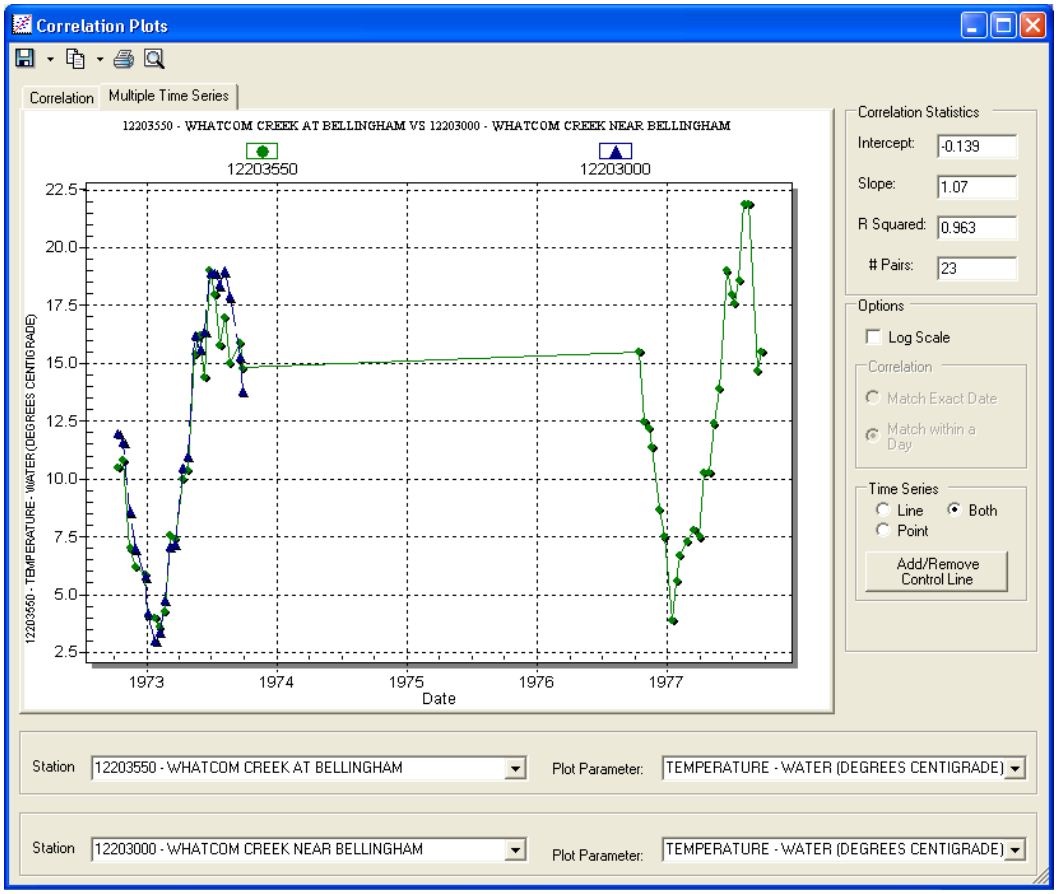
An option at the right of the form allows the user to control the way data pairs are selected for analysis. If “Match Exact Date” is selected, paired observations are included only if they share the same date *and* time. If “Match within a Day” is selected, data pairs are selected if they share the same date.




The Correlation Analysis form also includes the save, copy, print and undo zoom buttons for exporting the correlation plots. The function of these buttons is identical to those on the main Time Series Analyst form.

### *Multiple Time Series Analysis*

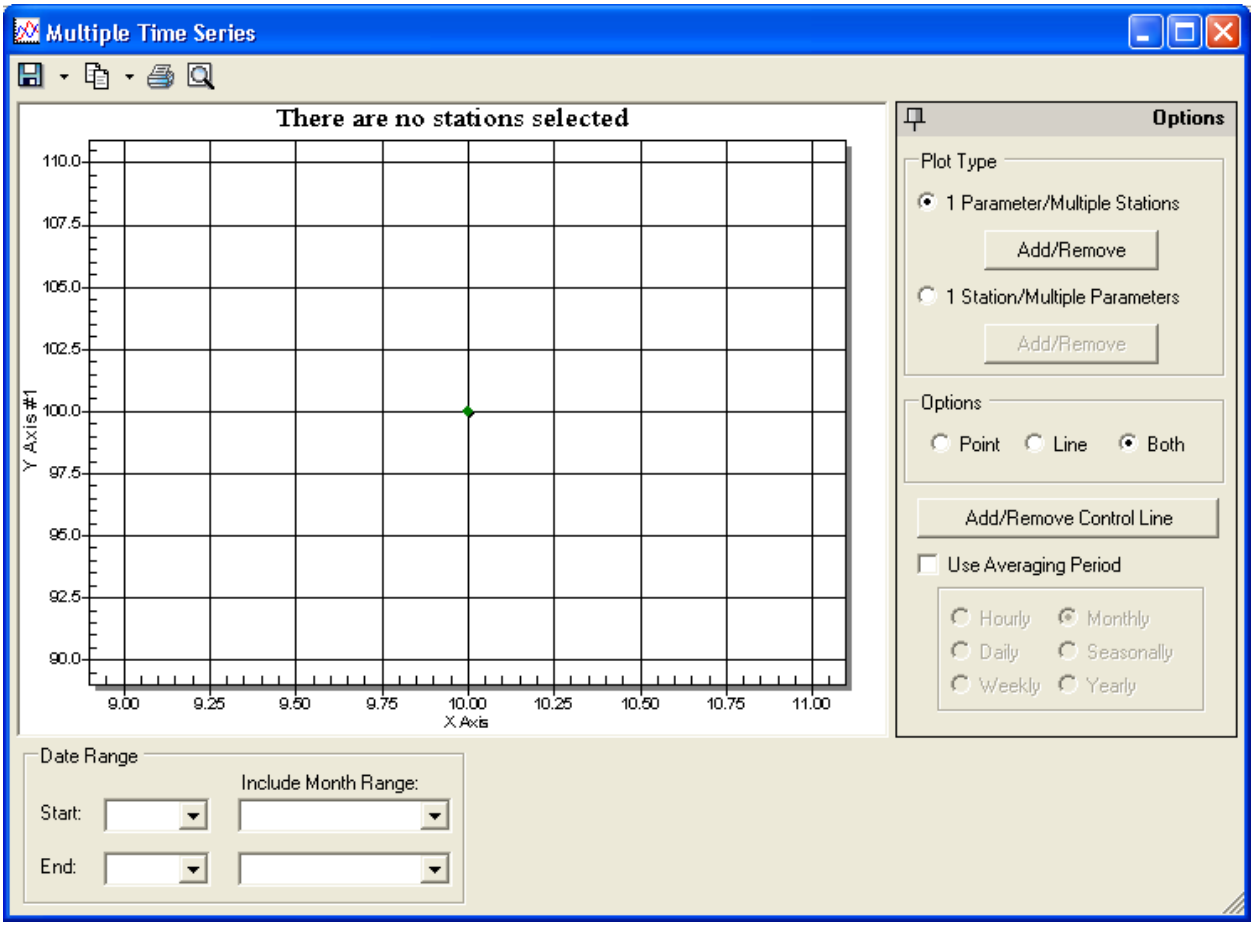
Click on the Multiple Time Series tab to view the sites and parameters chosen for the correlation plot as separate time series on the same plot as shown below. Notice that in the example, all paired sites are from before 1974. There is insufficient data for more correlative pairs.



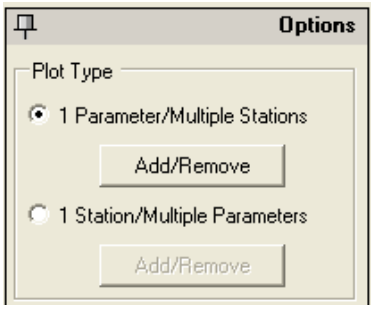

Formatting, display and exporting options can be accessed by right clicking on the plot window. These options are the same as those described for the main Time Series Analyst.

## Multiple Time Series Analysis

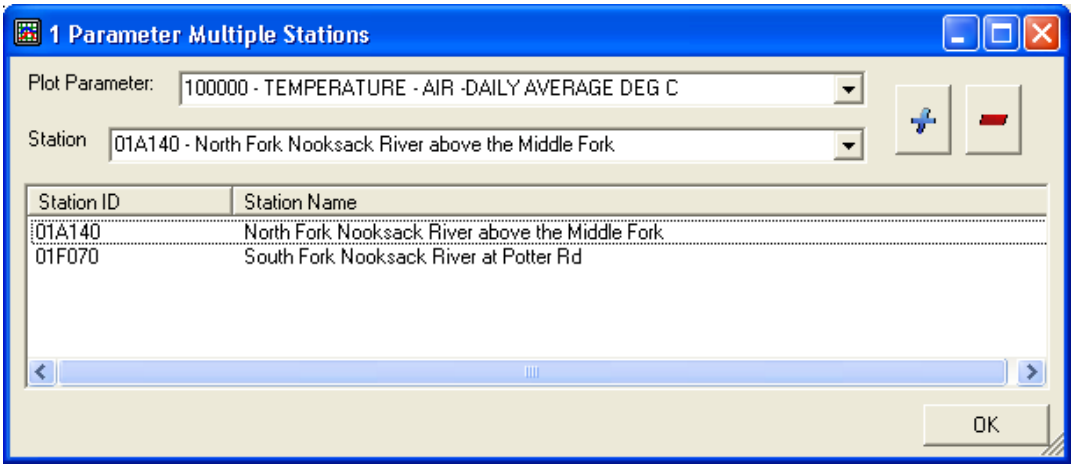
The Multiple Time Series Analysis tool allows the user to plot multiple time series of water quality on the same set of axes and is activated by clicking on the Multiple Time Series button on the toolbar. When the Multiple Time Series Analysis tool is activated, the window, shown below, will indicate that no stations are selected and the plot window will have no data displayed.



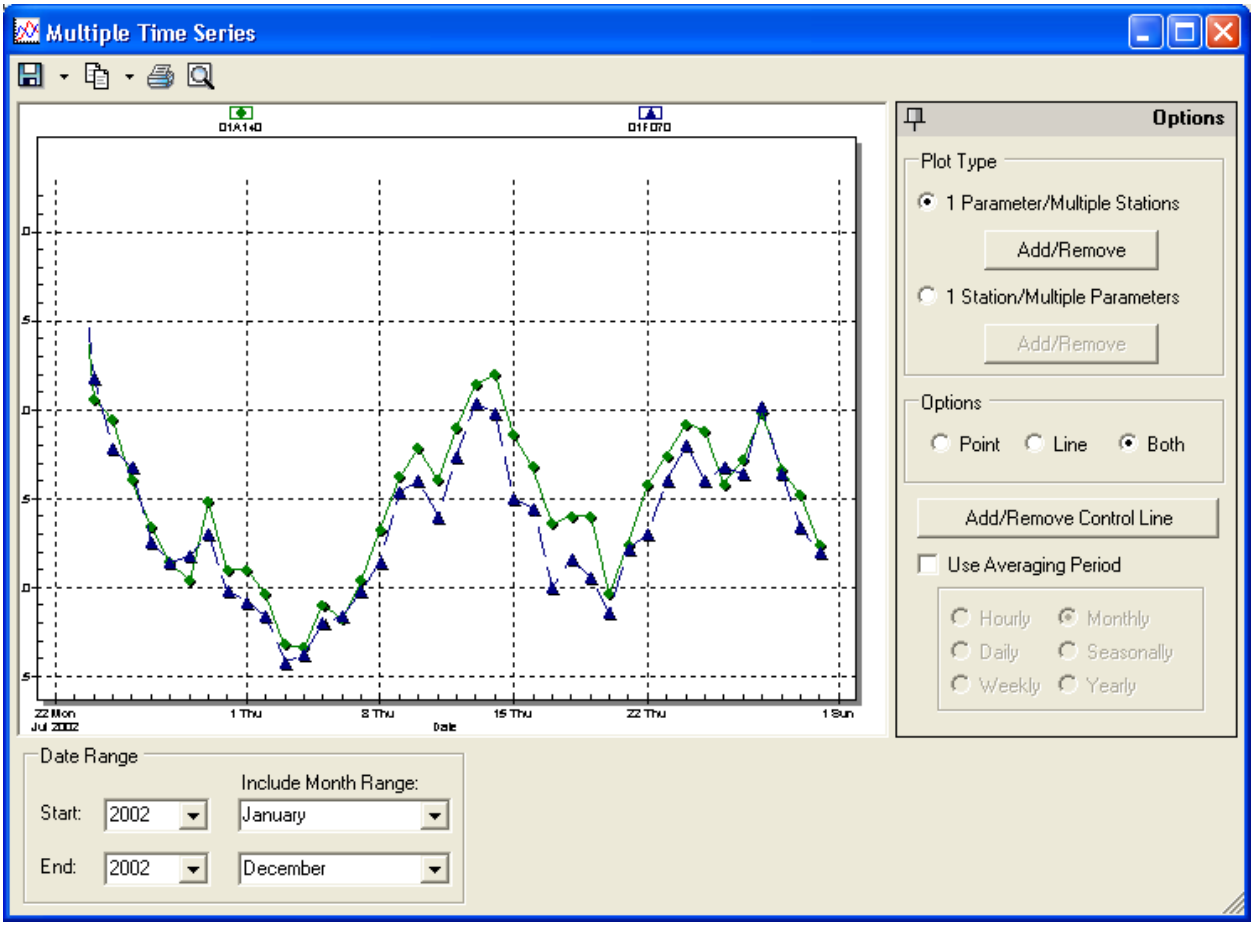

The user can choose to plot either one parameter at multiple stations or multiple parameters at a single station by clicking on the appropriate Plot Type option in the Options frame located on the right side of the form.




Station/Parameter combinations are added to the plot using the Add/Remove button under each plot type. An example of this form activated by the Add/Remove button is shown below.

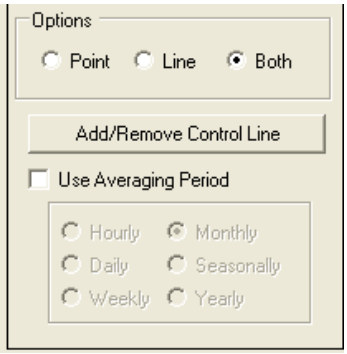

1 Parameter Multiple Stations
Plot Parameter:
100000 - TEMPERATURE - AIR -DAILY AVERAGE DEG C
Station
01A140 - North Fork Nooksack River above the Middle Fork
Station ID
Station Name
01A140
North Fork Nooksack River above the Middle Fork
01F070
South Fork Nooksack River at Potter Rd
OK

Station/Parameter combinations are selected using the pull down boxes at the top of the form. The button is clicked to add the combination to the plot. Clicking saves the selected Station/Parameter combination and updates the plot window.




Using the options on the right side of the form, the user can specify whether the time series are represented as points, lines, or both. In addition, the user can superimpose a

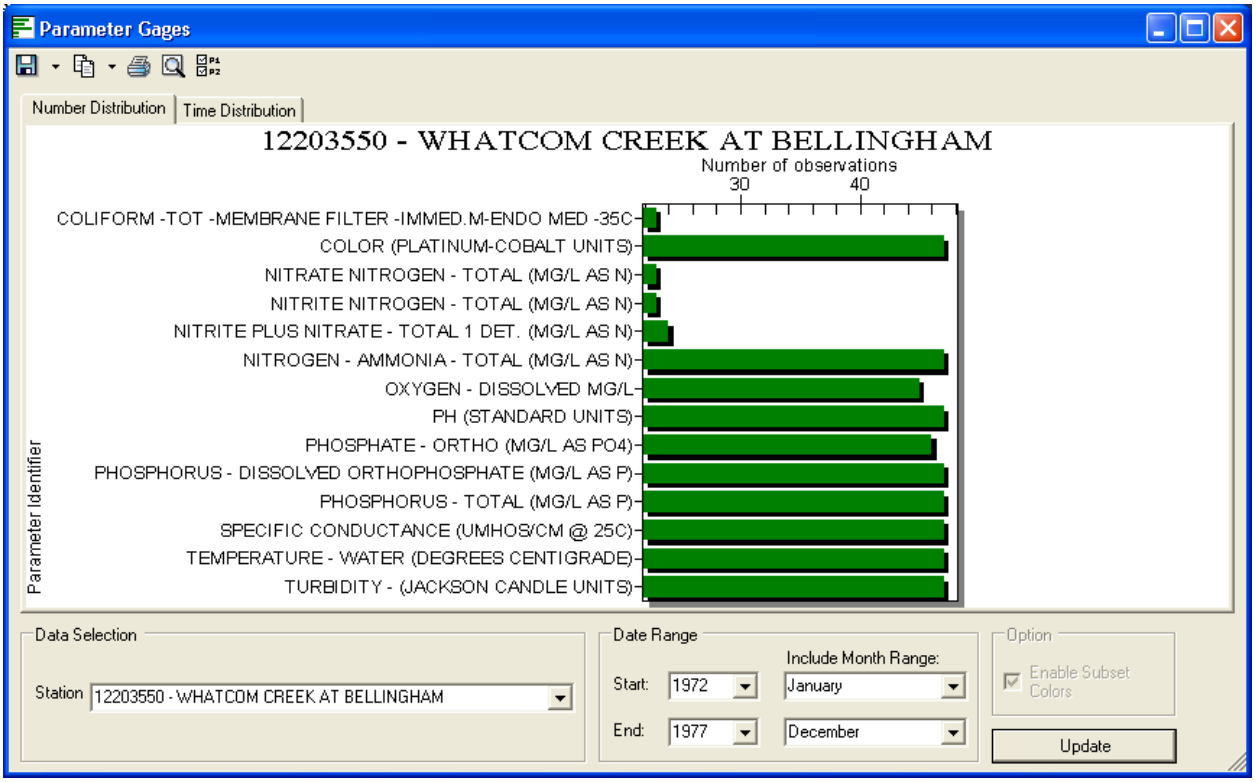
control line using the Add/Remove Control Line button as discussed in Statistical Summary, Data Sources, Plot Options, Plot Enhancements and Notes (in section for Plot Options tab). The "Use Averaging Period" option is available in the Multiple Time Series tool and is implemented in the manner discussed in the Plot Options tab section in the Time Series Analyst Options section above.




The Date Range fields at the bottom of the window can be used to modify the range of data shown in the plot as discussed in the Restricting Data Range section in the Time Series Analyst Options section above.

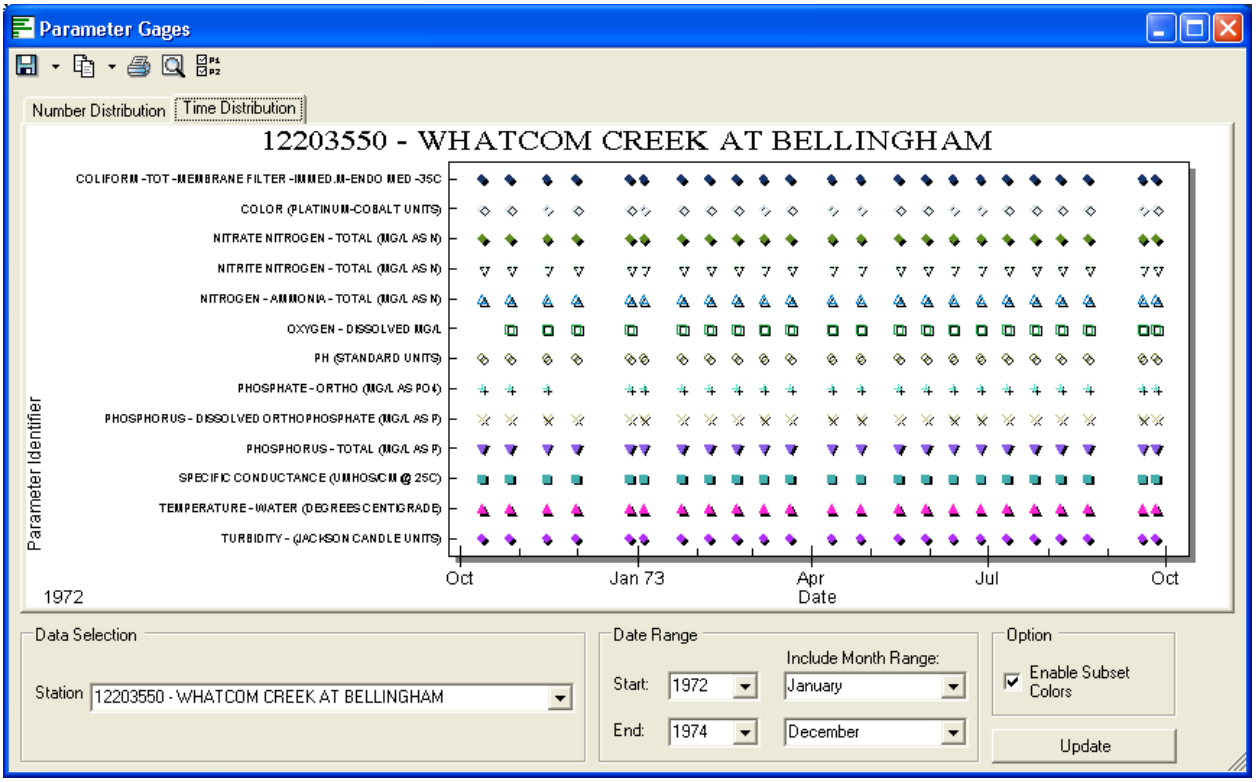
## Parameter Summary Analysis

The Parameter Summary Plots tool provides the user with plots of the number of observations for each parameter at each Water Quality station and the date distribution for each parameter at each station. This tool is accessed by clicking on the Parameter Summary Plots toolbar button. The following figure shows the Number Distribution plot generated by the Parameter Summary Plots tool.



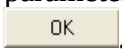

The plot shown above will automatically be generated for the selected Water Quality station in the Station pull down box on the main Time Series Analyst form. From this form, users can select and generate summaries for any of the water quality stations in the database by choosing stations from the Station pull down box. Water quality parameters are listed on the Y-axis, with the X-axis representing the number of water quality observations at the selected station.

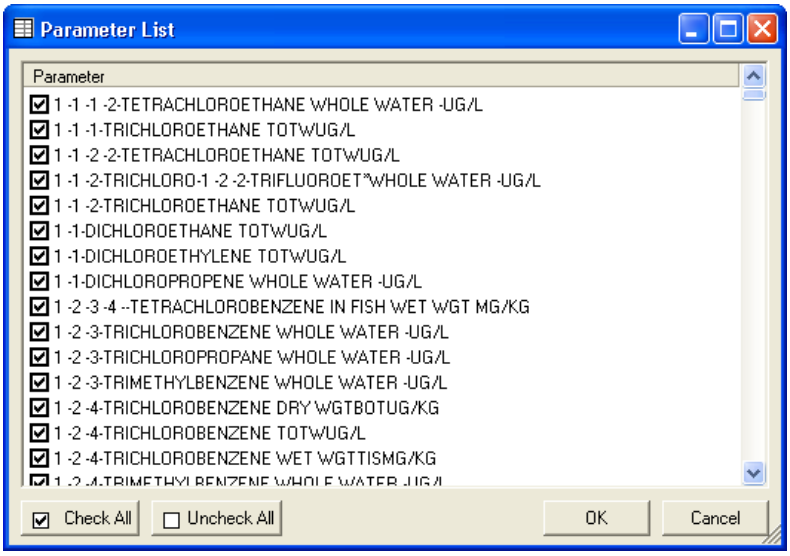
Select the Time Distribution tab to view the time distribution of water quality observations for each parameter at the selected station. Again, the water quality parameters are listed on the Y-axis but in this plot the X-axis represents the date an observation was made. Points on the plot represent an observation of water quality for a given parameter on a given date and give users an idea of the temporal distribution of the available water quality data.




The Date Range pull-down boxes show the range of dates represented in the plot. Users can limit this date range by selecting a new starting year and month and/or a new ending year and month. This allows users to identify water quality stations having data within specified time periods. It also provides users with information on the number of observations available within specific time periods. The Update button forces the plot to re-draw, ensuring that the plot represents the current settings in the Station and Date Range pull down boxes.

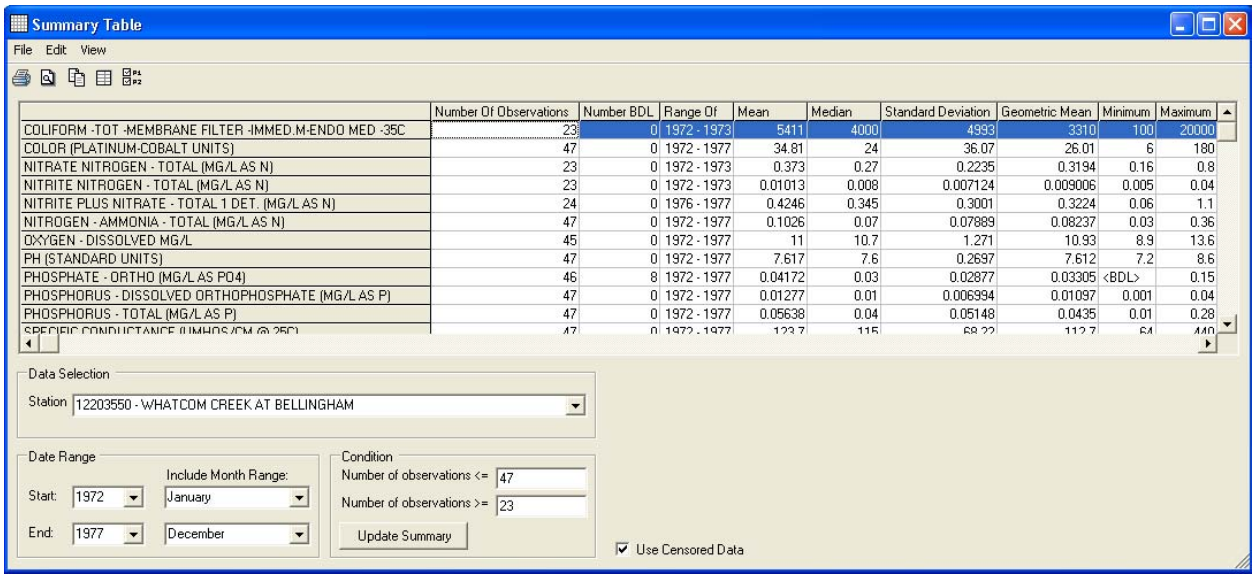
The Parameter Summary Plots form also includes the save, copy, and print buttons for exporting the plots that are generated. The function of these buttons is identical to those on the main Time Series Analyst form (refer to Plot Saving, Exporting, and Printing section above). An additional option at the bottom right of the form, Enable Subset Colors, allows the user to toggle between color and black & white symbols on the Time distribution plot. A complete set of plot options can be accessed by positioning the pointer on the plot window and clicking the right mouse button.

By default, all parameters for a station are shown in the above plots. The Select Parameters button at the top of the form allows the user to select a subset of water quality parameters from the list of all parameters in the water quality database. Select to access the form shown below. Use the check boxes to select/deselect the parameters of interest for the chosen station. When finished making selection, click  . Number Distribution and Time Distribution plots are automatically updated to display data only for the selected parameters.


Parameter List
Parameter
1 -1 -1 -2-TETRACHLOROETHANE WHOLE WATER -UG/L
1 -1 -1-TRICHLOROETHANE TOTWUG/L
1 -1 -2 -2-TETRACHLOROETHANE TOTWUG/L
1 -1 -2-TRICHLORO-1 -2 -2-TRIFLUOROET*WHOLE WATER -UG/L
1 -1 -2-TRICHLOROETHANE TOTWUG/L
1 -1-DICHLOROETHANE TOTWUG/L
1 -1-DICHLOROETHYLENE TOTWUG/L
1 -1-DICHLOROPROPENE WHOLE WATER -UG/L
1 -2 -3 -4 --TETRACHLOROBENZENE IN FISH WET WGT MG/KG
1 -2 -3-TRICHLOROBENZENE WHOLE WATER -UG/L
1 -2 -3-TRICHLOROPROPANE WHOLE WATER -UG/L
1 -2 -3-TRIMETHYLBENZENE WHOLE WATER -UG/L
1 -2 -4-TRICHLOROBENZENE DRY WGTBOTUG/KG
1 -2 -4-TRICHLOROBENZENE TOTWUG/L
1 -2 -4-TRICHLOROBENZENE WET WGTTISMG/KG
Check All
Uncheck All
OK
Cancel

## Statistical Summaries

The Parameter Summary Table option, activated by clicking on the Parameter Summary Table toolbar button provides tabular statistical summaries for data at selected water quality stations. The following figure shows an example tabular statistical summary.


Summary Table
File Edit View
Number Of Observations | Number BDL | Range Of | Mean | Median | Standard Deviation | Geometric Mean | Minimum | Maximum
COLIFORM -TOT -MEMBRANE FILTER -IMMED.M-ENDO MED -35C | 23 | 0 | 1972 - 1973 | 5411 | 4000 | 4993 | 3310 | 100 | 20000
COLOR (PLATINUM-COBALT UNITS) | 47 | 0 | 1972 - 1977 | 34.81 | 24 | 36.07 | 26.01 | 6 | 180
NITRATE NITROGEN - TOTAL (MG/L AS N) | 23 | 0 | 1972 - 1973 | 0.373 | 0.27 | 0.2235 | 0.3194 | 0.16 | 0.8
NITRITE NITROGEN - TOTAL (MG/L AS N) | 23 | 0 | 1972 - 1973 | 0.01013 | 0.008 | 0.007124 | 0.009006 | 0.005 | 0.04
NITRITE PLUS NITRATE - TOTAL 1 DET. (MG/L AS N) | 24 | 0 | 1976 - 1977 | 0.4246 | 0.345 | 0.3001 | 0.3224 | 0.06 | 1.1
NITROGEN - AMMONIA - TOTAL (MG/L AS N) | 47 | 0 | 1972 - 1977 | 0.1026 | 0.07 | 0.07889 | 0.08237 | 0.03 | 0.36
OXYGEN - DISSOLVED MG/L | 45 | 0 | 1972 - 1977 | 11 | 10.7 | 1.271 | 10.93 | 8.9 | 13.6
PH (STANDARD UNITS) | 47 | 0 | 1972 - 1977 | 7.617 | 7.6 | 0.2697 | 7.612 | 7.2 | 8.6
PHOSPHATE - ORTHO (MG/L AS PO4) | 46 | 8 | 1972 - 1977 | 0.04172 | 0.03 | 0.02877 | 0.03305 | <BDL> | 0.15
PHOSPHORUS - DISSOLVED ORTHOPHOSPHATE (MG/L AS P) | 47 | 0 | 1972 - 1977 | 0.01277 | 0.01 | 0.006994 | 0.01097 | 0.001 | 0.04
PHOSPHORUS - TOTAL (MG/L AS P) | 47 | 0 | 1972 - 1977 | 0.05638 | 0.04 | 0.05148 | 0.0435 | 0.01 | 0.28
Data Selection
Station 12203550 - WHATCOM CREEK AT BELLINGHAM
Date Range
Include Month Range:
Start: 1972 January
End: 1977 December
Condition
Number of observations <= 47
Number of observations >= 23
Update Summary
Use Censored Data

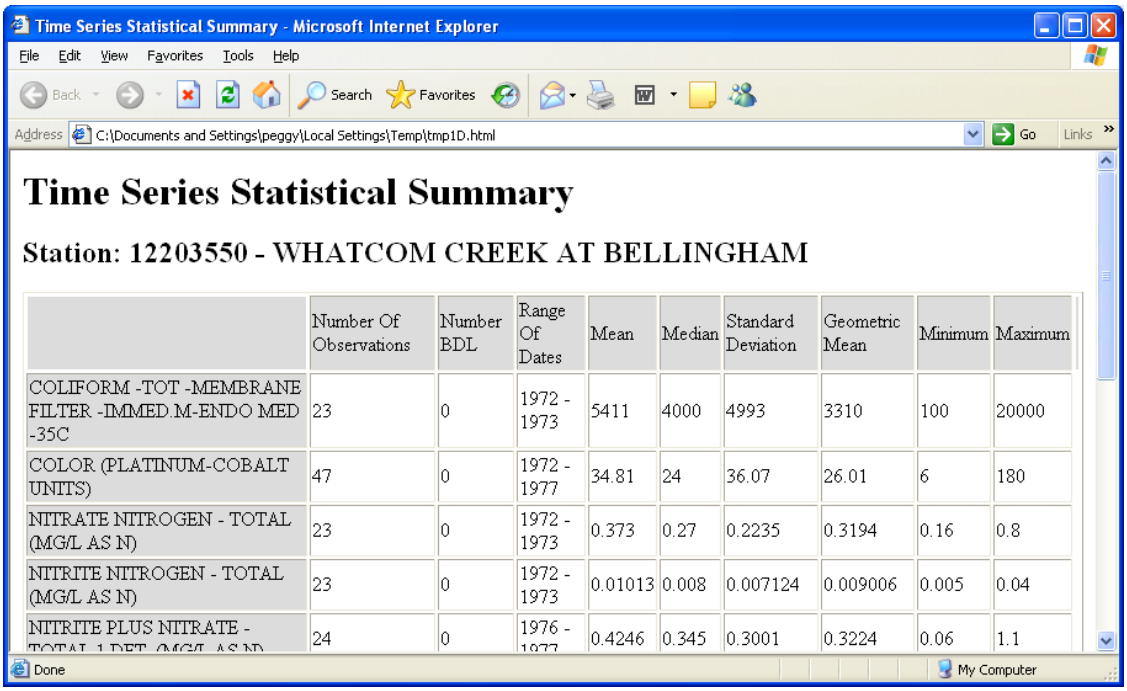
The table provides descriptive statistics (listed as column headings at the top of the table) for each water quality parameter (listed as row headings on the left side of the table) for the selected station. Again, stations for analysis are selected using the Station pull down box, and the date range for analysis can be controlled using the Date Range pull down boxes. Conditions can also be set using the Condition fields to include only parameters

for which a desired number of observations exist. The Update Summary button applies the changes made to the Station, Data Range, and Condition fields.

Similar to the Parameter Summary Analysis (discussed previously), the user can choose the Select Parameters button to specify a subset of water quality parameters to include in the table. Censored data may also be included in the calculation of the descriptive statistics by clicking on the Use Censored Data check box.

The rows in the statistical summary table can be selected and copied for pasting into reports or other documents. Selecting records is done by clicking on records in the table. Once selected, rows can be copied by clicking on the copy button. The copied records can then be pasted into a report or other document.

The print button automatically generates an HTML format report of all of the information shown in the table that can be sent directly to a printer or posted on a web page. This report can be previewed in a web browser by clicking on the Preview Report button.

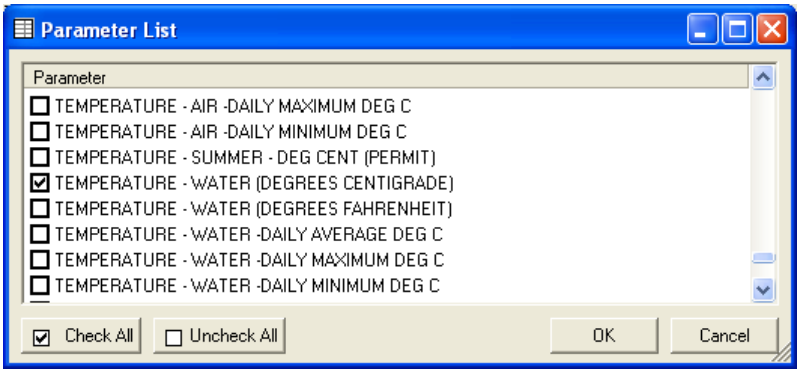



**Time Series Statistical Summary**

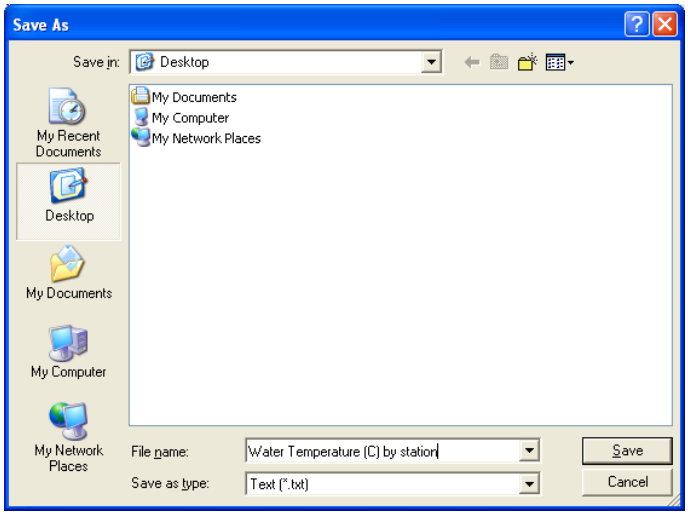
**Station: 12203550 - WHATCOM CREEK AT BELLINGHAM**

| | Number Of Observations | Number BDL | Range Of Dates | Mean | Median | Standard Deviation | Geometric Mean | Minimum | Maximum |
|---|---|---|---|---|---|---|---|---|---|
| COLIFORM -TOT -MEMBRANE FILTER -IMMED M-ENDO MED -35C | 23 | 0 | 1972 - 1973 | 5411 | 4000 | 4993 | 3310 | 100 | 20000 |
| COLOR (PLATINUM-COBALT UNITS) | 47 | 0 | 1972 - 1977 | 34.81 | 24 | 36.07 | 26.01 | 6 | 180 |
| NITRATE NITROGEN - TOTAL (MG/L AS N) | 23 | 0 | 1972 - 1973 | 0.373 | 0.27 | 0.2235 | 0.3194 | 0.16 | 0.8 |
| NITRITE NITROGEN - TOTAL (MG/L AS N) | 23 | 0 | 1972 - 1973 | 0.01013 | 0.008 | 0.007124 | 0.009006 | 0.005 | 0.04 |
| NITRITE PLUS NITRATE - | 24 | 0 | 1976 - | 0.4246 | 0.345 | 0.3001 | 0.3224 | 0.06 | 1.1 |

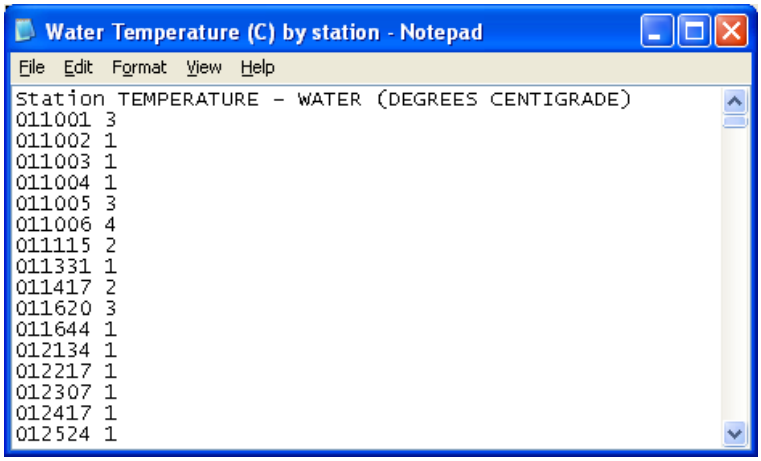
To create a summary of records by parameter for all stations, click on. This will bring up the Parameter List as shown below. Select the parameters of interest using the check boxes. Click OK when finished.


Parameter List
Parameter
TEMPERATURE - AIR -DAILY MAXIMUM DEG C
TEMPERATURE - AIR -DAILY MINIMUM DEG C
TEMPERATURE - SUMMER - DEG CENT (PERMIT)
TEMPERATURE - WATER (DEGREES CENTIGRADE)
TEMPERATURE - WATER (DEGREES FAHRENHEIT)
TEMPERATURE - WATER -DAILY AVERAGE DEG C
TEMPERATURE - WATER -DAILY MAXIMUM DEG C
TEMPERATURE - WATER -DAILY MINIMUM DEG C
Check All
Uncheck All
OK
Cancel

The user will be prompted to supply a file name and location for the output file containing the summary of records by parameter.

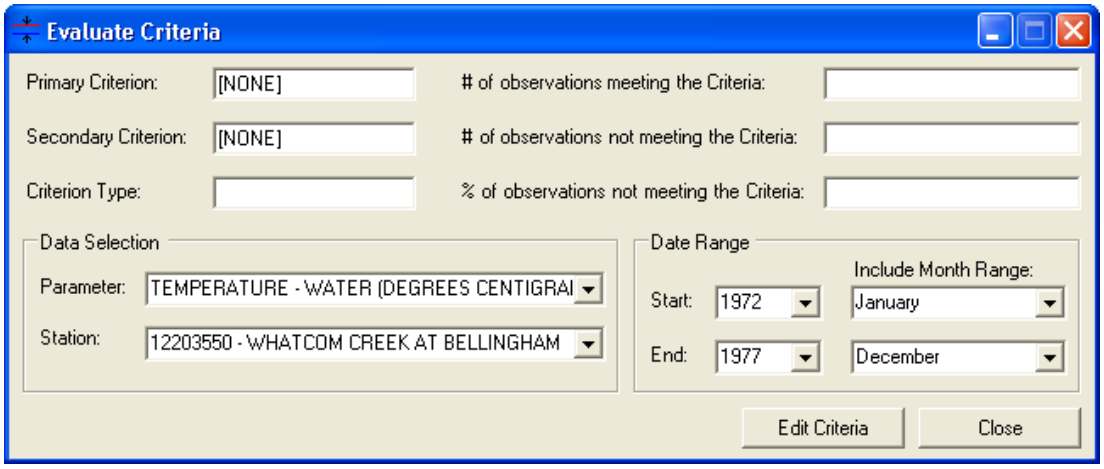



The resulting output file lists stations by identification number and the number of observations of the specified parameter that were made at each station. An example output file is shown below.



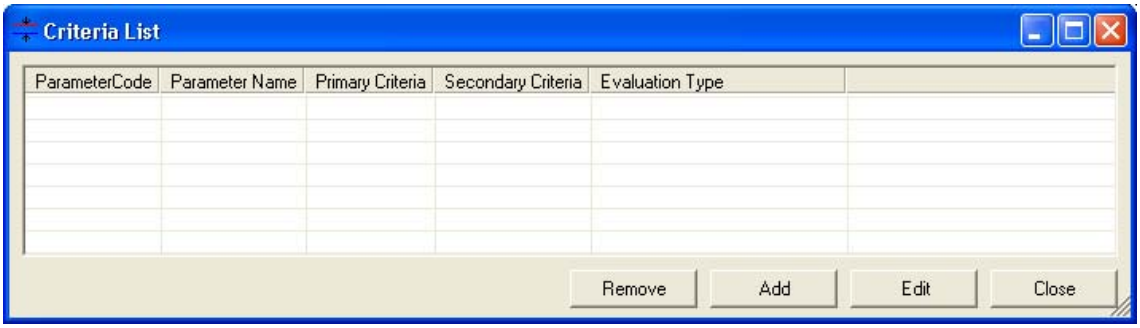
## Evaluate Criteria

The Evaluate Criteria option is activated by clicking on the Evaluate Criteria button on the toolbar.

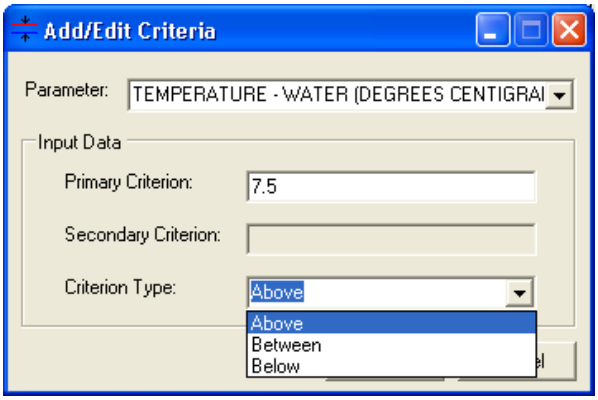



For any given parameter at any station, a user may find the number of observations that meet and do not meet specified criteria. In addition, the percentage of observations that do not meet the criteria is also supplied.

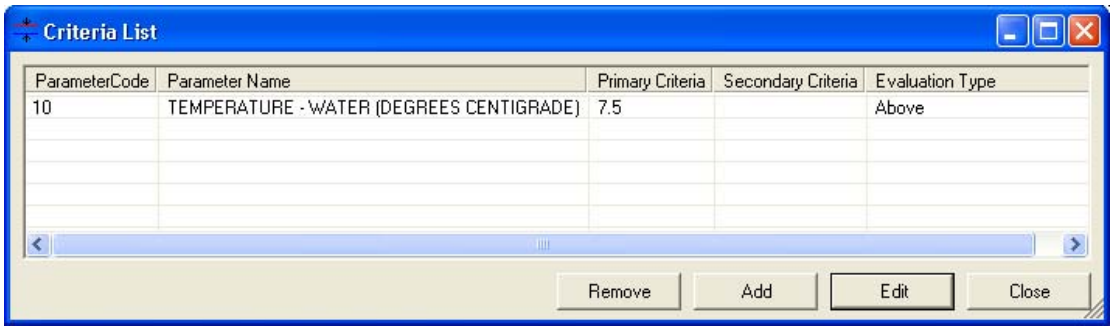
Select a station (or group if Station Group is activated) from the Station pull down box and the parameter of interest from the Parameter pull down box. The user may restrict the data range by changing the dates in the Date Range pull down boxes. Select Edit Criteria on the Evaluate Criteria form to set the criteria for evaluation of the specified parameter. A blank Criteria List window will appear.




Click on Add to access the Add/Edit Criteria form.

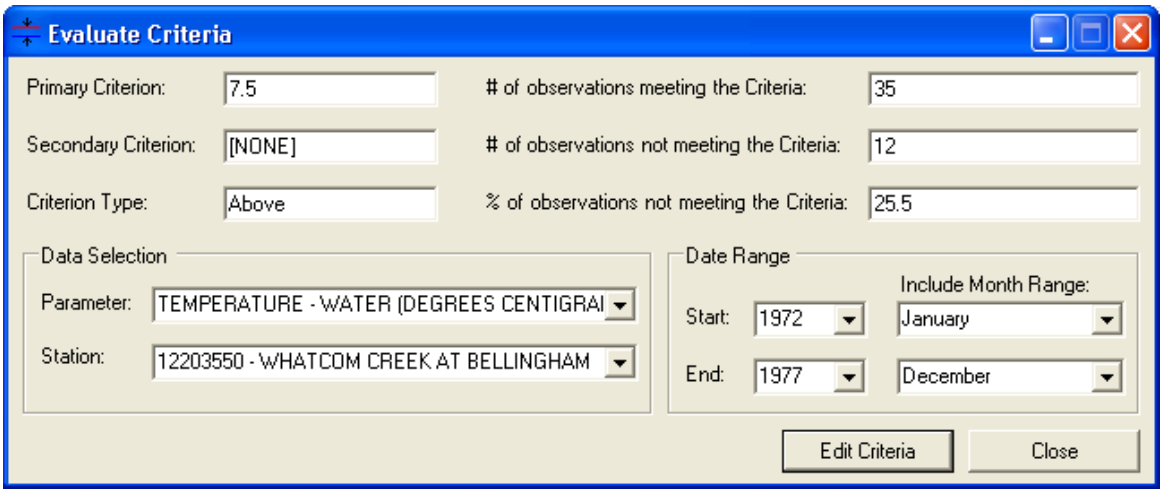



Parameter pull down box will be populated by the selection made in the initial Evaluate Criteria window. Enter Primary Criterion and Secondary Criterion (optional) in the appropriate spaces. Select a Criterion Type from the pull down box and click OK. The Criteria List window now appears as follows.



To remove or edit criteria on the list, use the mouse to select the parameter from the list and click Remove or Edit, as appropriate. More parameters can be added from this form by clicking on the Add button, however it is advisable to click on Close and go to the initial Evaluate Criteria form. If one attempts to add a new parameter for criteria evaluation by selecting Add in the Criteria List window, all available parameters from the entire database are listed in the Parameter pull down box in the Add/Edit Criteria form. If however, the user proceeds from the initial Evaluate Criteria form, the parameters listed in that Parameter pull down box are only those that are applicable to the station specified in the Station pull down box.

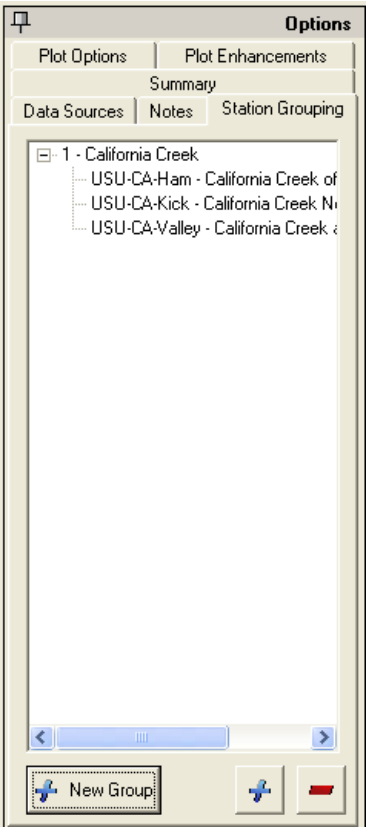
The result of the Evaluate Criteria operation used for this example is shown below. The results indicate that 35 observations are above 7.5C and 12 observations are below 7.5C. Also, 25.5% of the observations did not meet the criteria (were below 7.5C).




## Station Grouping

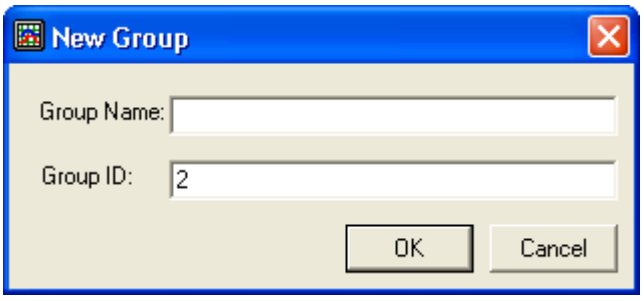
The Station Grouping option is activated by clicking on the Station Grouping toolbar button . This option allows the user to group data from multiple water quality stations for analysis. This is particularly useful when two or more water quality stations are located in the same vicinity. The data collected at these stations can be combined for the purposes of generating plots and statistical analyses. An example would be creating plots and summaries for all data within a subwatershed or on a particular stream reach. All stations within the subwatershed or on the stream reach can be grouped so that plots and summaries generated would represent all available data within that watershed or stream reach.

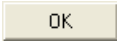
Clicking on the Station Grouping toolbar button activates the Station Grouping option and adds an additional tab called Station Grouping to the Options window on the right side of the Time Series Analyst. In the tab shown below, a user can define a station group for data analysis.

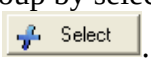


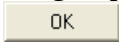

Clicking on

activates the New Group Form, shown below, in which a name may be specified for the new group that is to be created. The Group ID field is automatically populated.

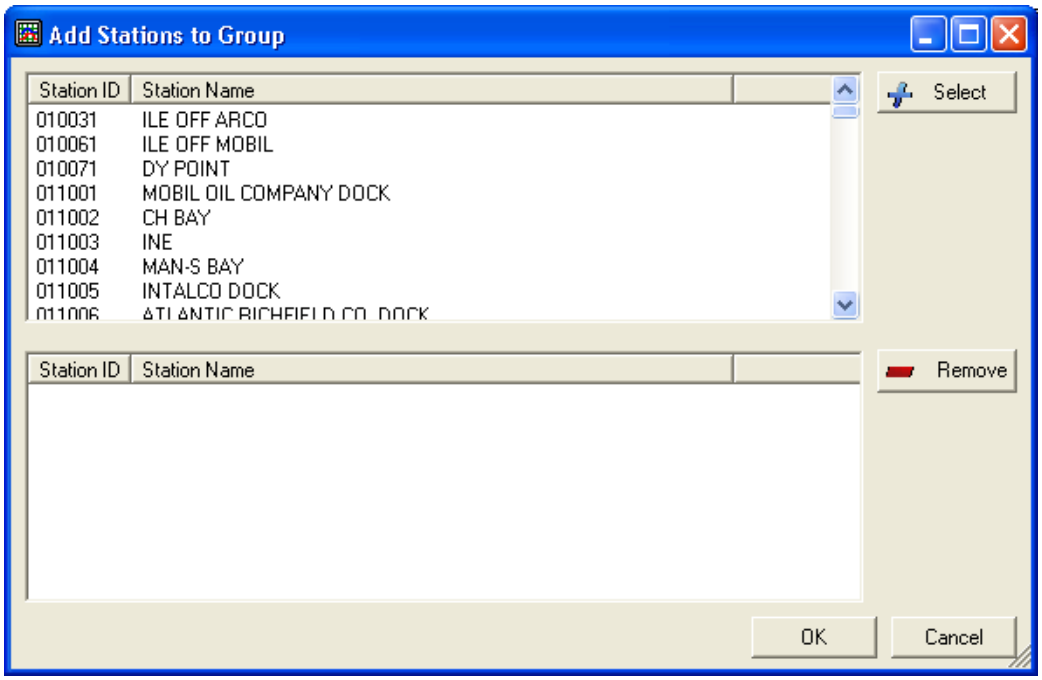


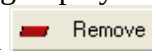

When the user enters a Group Name and clicks 

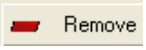
on the New Group form, the station group is created and the Add Stations to Group form appears. A station is added to the group by selecting it from the list of stations in the upper window of the form and clicking 

. The selected station is added to the list of stations to be included in the group, which will be shown in the lower window of the Add Stations form. Click 

when finished adding stations to the group.




The Station Grouping tab now shows the newly created group and the stations that have been added to it. Stations can easily be removed from a group by selecting them in the Station Grouping tab and clicking on the Remove button 

at the bottom of the Station Grouping tab. Stations can be added or removed from any existing station group by simply using the Add Station and Remove Station buttons at the bottom of the Station Grouping tab. Station groups may be deleted by selecting the group name in the Station Grouping tab and clicking on the 

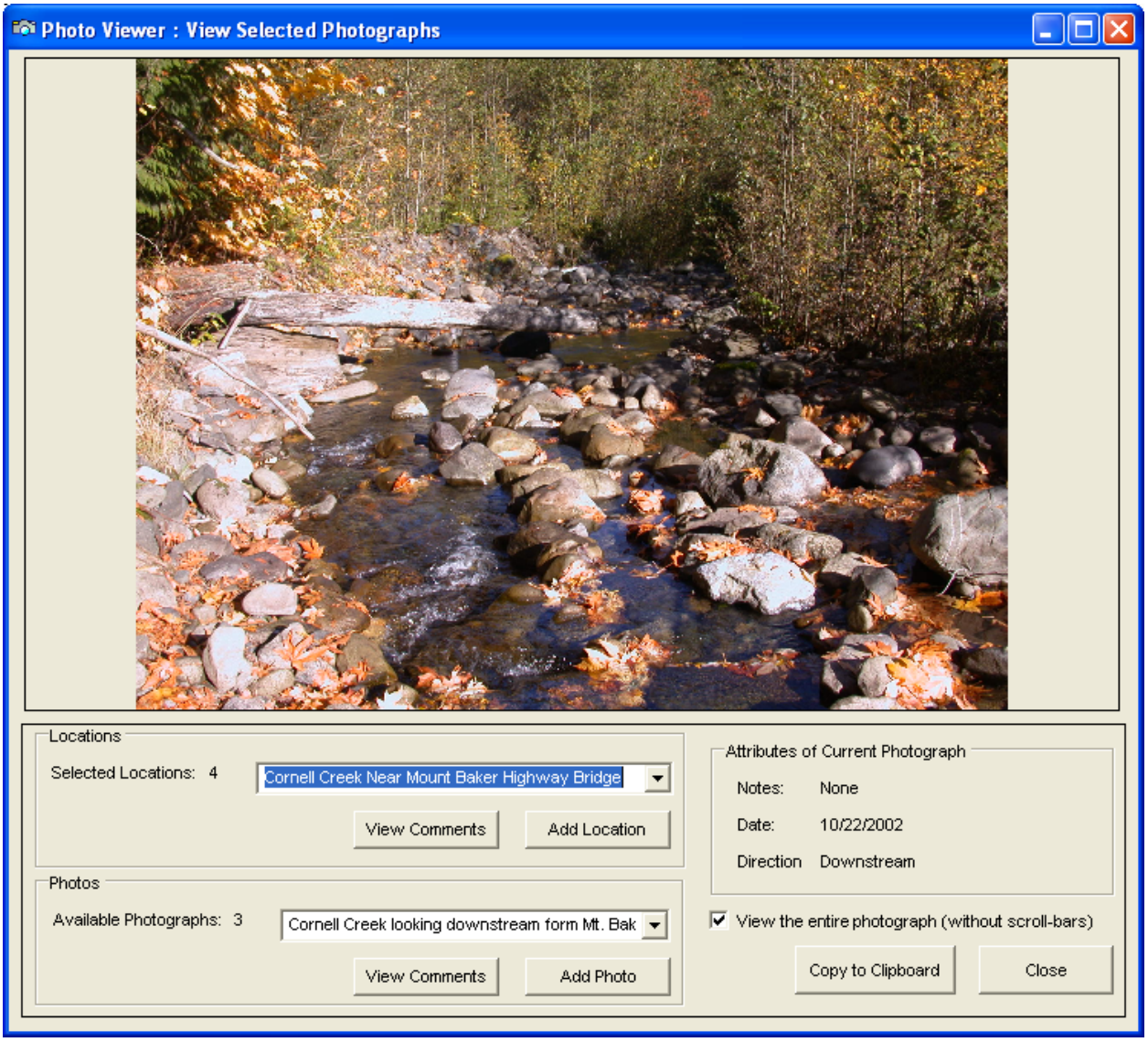
button.

With the Station Grouping enabled, the user may now perform any of the analyses already discussed for single stations. Clicking on the Station Grouping toolbar button again will disable the Station Grouping option and switch the Time Series Analyst back to single station mode.

The Undo Zoom button can be used with any of the graphing options to return the plot window to the full extent.

# Photo Viewer Plug-in

The Photo Viewer Plug-in to the DSS allows users to view digital photographs taken at locations throughout the watershed. A database of digital photographs and their descriptions (i.e., location, date, photo description, etc.) is linked to a geographic coverage showing the locations at which the photographs were taken. To access this tool, press the button on the DSS toolbar. If this button is not shown on the toolbar, go to the Plug-ins menu and select Photo Viewer. The following is an example of the Photo Viewer form.

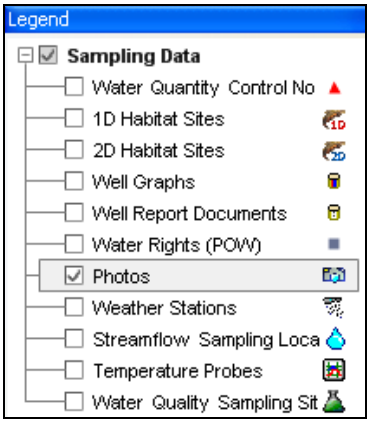


## Viewing Photographs

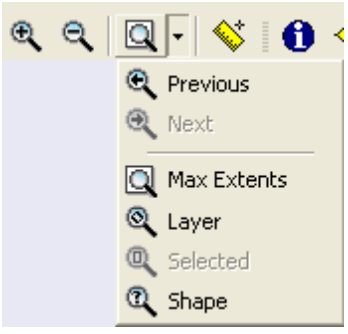
The available photographs in the watershed can be viewed by either clicking on and selecting a Photo Viewer point or group of Photo Viewer points from the map or by clicking on the Photo Viewer Toolbar button . Selecting points from the map loads the available photographs for the selected locations into the Photo Viewer, whereas clicking on the Photo Viewer toolbar button loads all available photographs in the entire database into the Photo Viewer.

### *Selecting Photo Viewer Points from the Map*

First make sure that the Photos theme is turned on (checked) and active (selected).



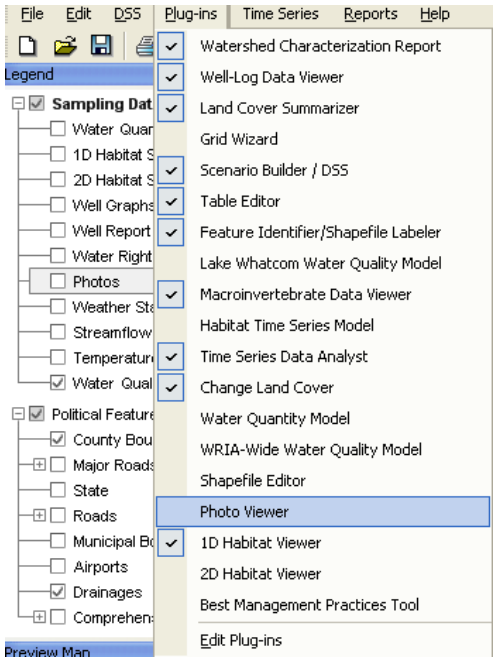

Next, use the Zoom and Pan Tools to locate the area of interest on the map. Additional Zoom tools are accessed by clicking on the black arrow as shown.




Using the select tool , points can be selected by either clicking on individual stations or by clicking and dragging a box around one or several locations. Selecting points from the Photo points theme will load the photographs for those selected locations into the Photo Viewer form shown above.

### *Viewing Photographs via the Photo Viewer Toolbar Button*

As an alternative to selecting points using the map, clicking the Photo Viewer toolbar button opens the Photo Viewer and loads the entire database of photographs into the Photo Viewer form shown above. If this button is not available on the toolbar, select the Plug-ins menu and choose Photo Viewer as shown below.




This will add the photo viewer button to the toolbar. By pressing this button, users can browse the entire database of photographs without interacting with the map.

## Photo Viewer Options

Several options are available on the Photo Viewer form. These options are described in the following sections.

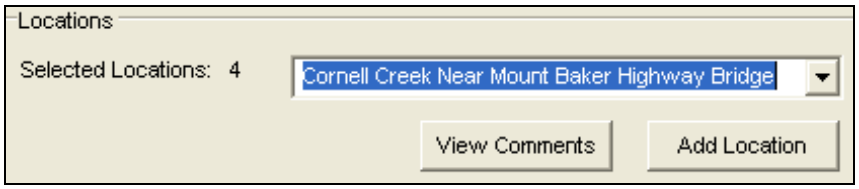
### *Photograph View Window*

When selected

scales the current photograph to fit in the view window (as shown in the example at the beginning of this section above) regardless of the original size of the digital image. When this option is deselected, (unchecked) the photo is shown in its original size and resolution. The user is able to use scroll bars along the bottom and side of the view window to scroll from side to side and top to bottom to view the entire photograph.

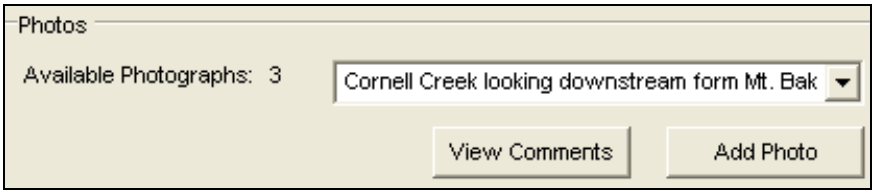
### *Location Selection*

Users can select from the list of available locations by using the location pull down box in the Locations section of the Photo Viewer window. The location pull down box contains a list of all selected locations for which photographs are available. The number of selected locations is shown next to the location pull down box. If points were selected from the map, the pull down box will list the selected locations only. If the Photo Viewer was launched by clicking on the Photo Viewer toolbar button, the location pull down box will list all of the photographic locations in the database.




### *Photograph Selection*

When more than one photograph was taken at a location, the pull down box in the Photos section of the Photo Viewer window allows the user to select from all photographs available for a location. The number of available photographs for the selected location is shown next to the pull down box.




### *Photograph Attributes*

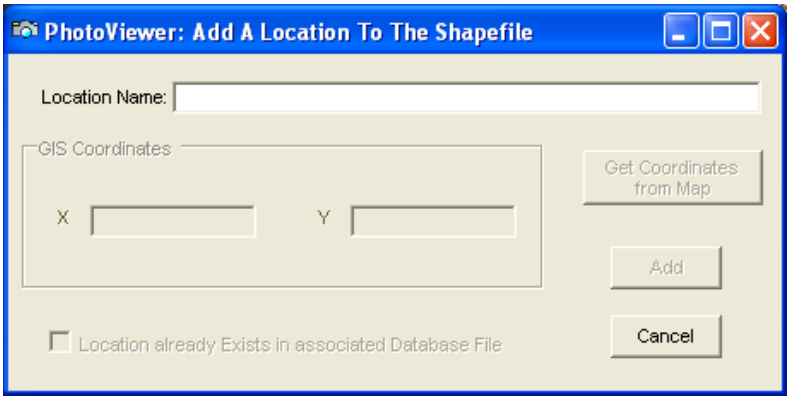
The attributes of the current photograph are shown on the Photo Viewer form. Attributes include the date the photograph was taken, the direction the camera was facing when taking the photograph and any brief notes accompanying the photograph.

### *Copy Photograph to Clipboard*

Click on Copy to Clipboard to copy the photograph that is currently in the photograph view window to the clipboard so it can be pasted into a report or other document for printing purposes. For best results, deselect (uncheck) View the entire photograph (without scroll-bars) to view the photograph at its original resolution before copying to the clipboard. The photo size will be the same when copied from the clipboard into a document whether the box is checked or not.

### *Add Location*

To add a new location to the Photo Viewer database, click on Add Location in the Locations section of the Photo Viewer form. The form shown below will appear.

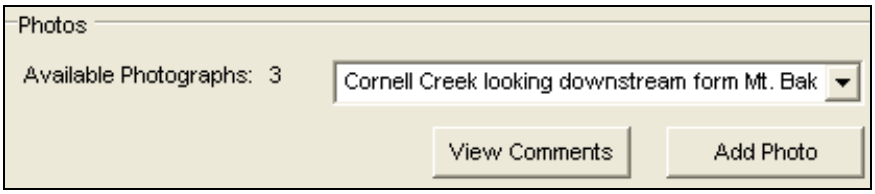



**Note:** The check box at the bottom of the form allows the user to specify whether the new location is already in the Photo Viewer database. Photo Viewer is designed so that it is possible to have locations in the Photo Viewer database that are not in the geographic coverage of locations. Users can add photos to the database, even if they don't have GPS coordinates or a shapefile of the locations at which they were taken. The above form can be used to specify the coordinates of the location so that it will be added to the geographic coverage of photograph locations.

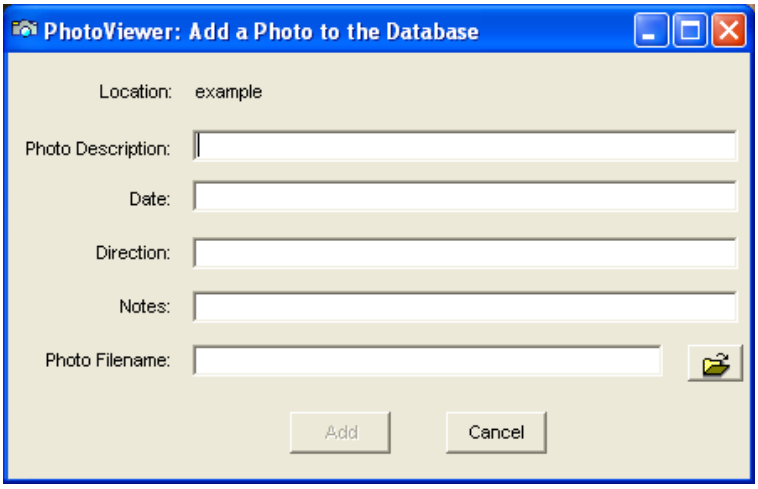
Enter the new location name in the Location Name window and the rest of the form will become activated. X and Y coordinates (i.e., GPS coordinates) can be entered manually or they can be entered automatically by clicking on the map. If coordinates are to be specified by using the map, click on Get Coordinates from Map. The map window in the DSS interface will appear. Click on the map location of the photograph and the X and Y coordinate fields will be populated automatically. Click Add to add the new location to the database and to the geographic coverage of photograph locations. The "Add a Photo to the Database" form, described in the following section, will appear so that a photo can be added to the new location that was just created.

### *Add Photo*

The Photo Viewer Plug-in allows users to add photographs to a new location that has just been created or to add additional photographs to locations that are already in the Photo Viewer database. To add a photo to the database, click on the Add Photo button in the Photographs section of the Photo Viewer form (below).



The "Add a Photo to the Database form (shown below) will appear. This same form appears automatically after adding a photograph location to the database as described in the previous section titled Add Location.

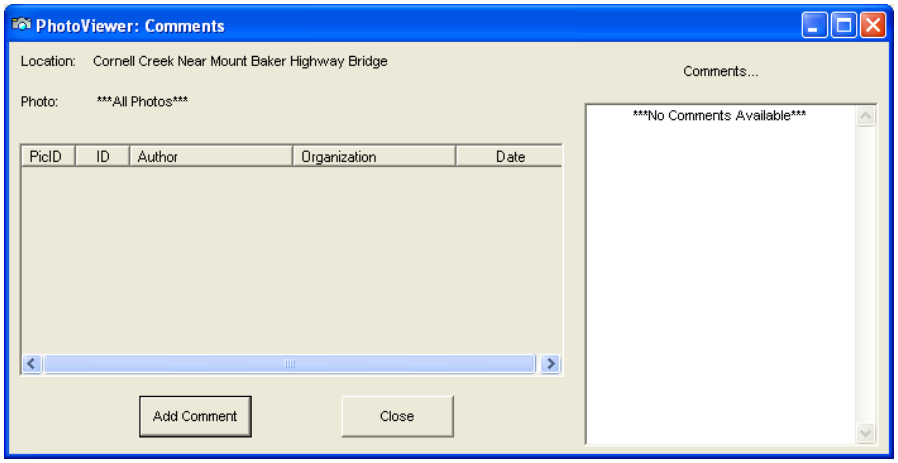



The fields in the Add Photo form must be filled out with the descriptive information about the photograph that is to be added. Once the descriptive fields are filled out, the user can browse to the location of the digital photograph on disk using the button to the right of the "Photo Filename" field to access the form shown below. Photograph file formats supported by Photo Viewer include jpeg and bitmap.

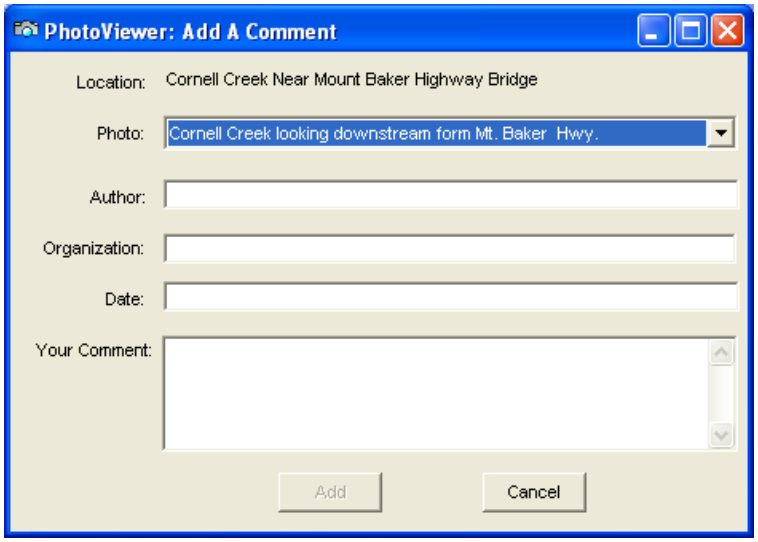
### *View Comments*

The view comments options in the Locations and Photos sections of the Photo Viewer allow users to view and add anecdotal information about locations and photographs to the Photo Viewer database. Clicking on

under the selected location will show all available comments for the currently selected location. Clicking on the View Comments button under the selected photograph will show all available comments for the selected photograph. The comments form is shown below.



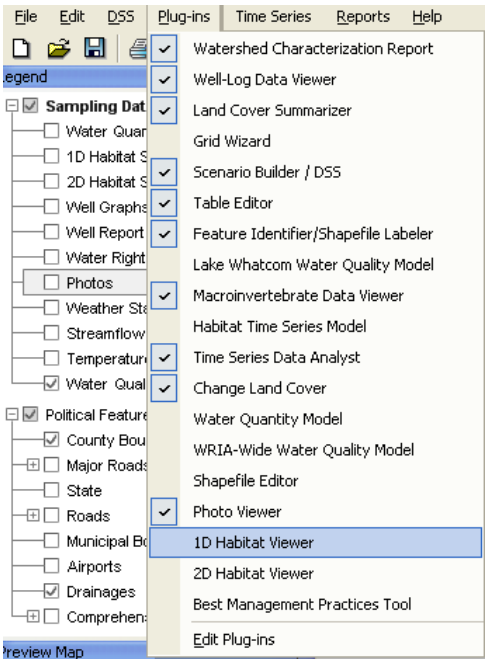

Users can add comments to the database by clicking on Add Comment. The following form will open.



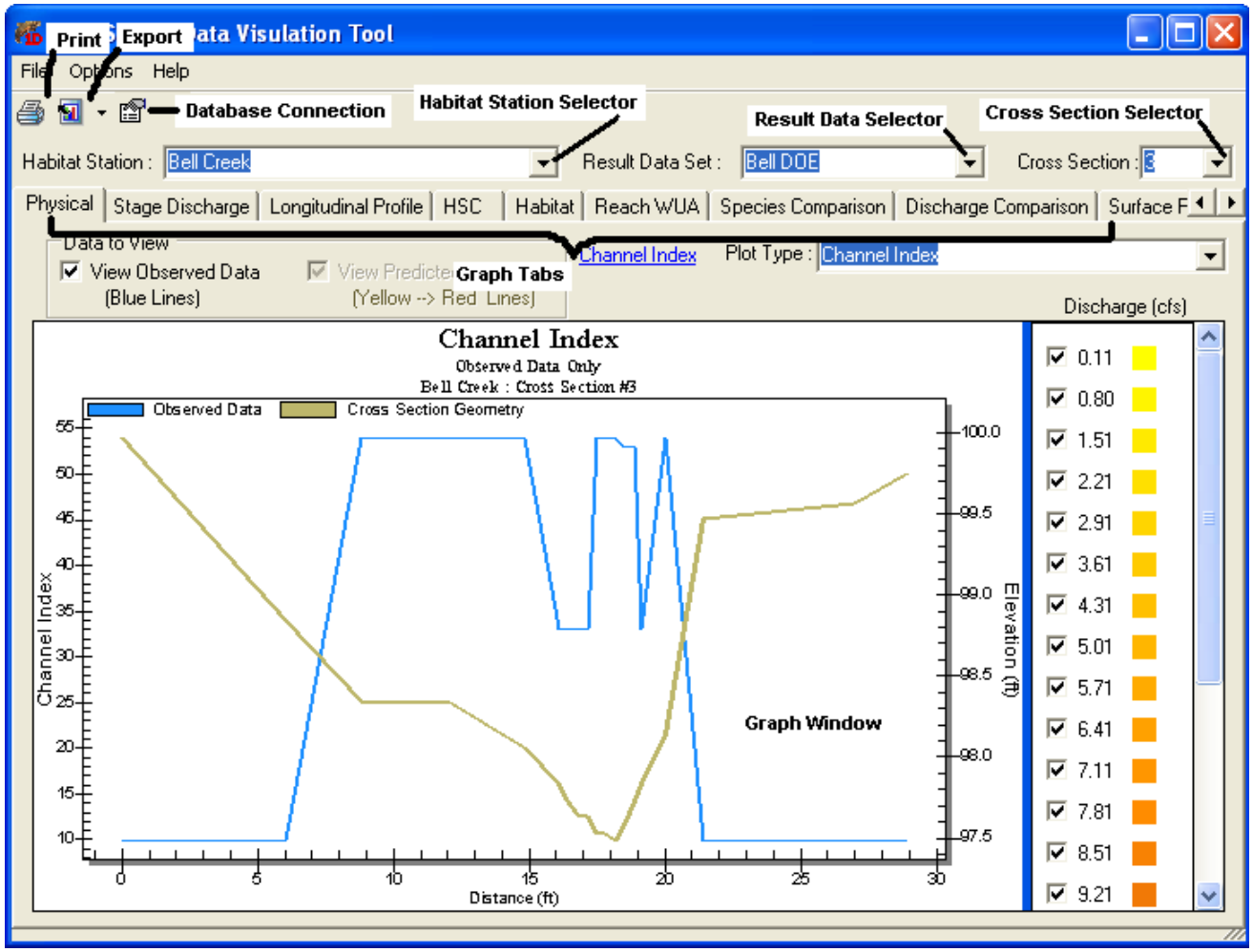

By filling in the blanks on the form and clicking Add, the comment is added to the database and will show up in the Comments form. It should be noted that the location field is automatically populated with the location currently selected in the Photo Viewer. The user must select the photo to which the new comment should be associated in the **Photo** pull down box. The Add button will become active when an author is entered in the Author box.

# 1D Habitat Viewer Plug-in

The 1D Habitat Viewer plug-in allows you to view and query information on the effects to fish habitat of different stream flow regimes. It supports both PHABSIM 1D modeling and Rapid Assessment modeling output. This plug-in is launched by clicking on the Phabsim 1-D Visualization button on the toolbar, . If this button is not available on the toolbar, click on the Plug-ins menu on the menu bar and select 1D Habitat Viewer from the list as shown below.




This will add the 1D Habitat Viewer button to the toolbar. When the toolbar button is clicked, the following form will appear. In the figure below, the different parts of the form are labeled for reference. The following sections describe each part of the viewer.


Print
Export
ata Visulation Tool
File
Options
Help
Database Connection
Habitat Station Selector
Result Data Selector
Cross Section Selector
Habitat Station :
Bell Creek
Result Data Set :
Bell DOE
Cross Section :
3
Physical
Stage Discharge
Longitudinal Profile
HSC
Habitat
Reach WUA
Species Comparison
Discharge Comparison
Surface F
Graph Tabs
Data to View
View Observed Data
(Blue Lines)
View Predicte
(Yellow --> Red Lines)
Channel Index
Plot Type :
Channel Index
Discharge (cfs)
Channel Index
Observed Data Only
Bell Creek : Cross Section #3
Observed Data
Cross Section Geometry
Graph Window
Channel Index
Elevation (ft)
Distance (ft)
0.11
0.80
1.51
2.21
2.91
3.61
4.31
5.01
5.71
6.41
7.11
7.81
8.51
9.21

4510

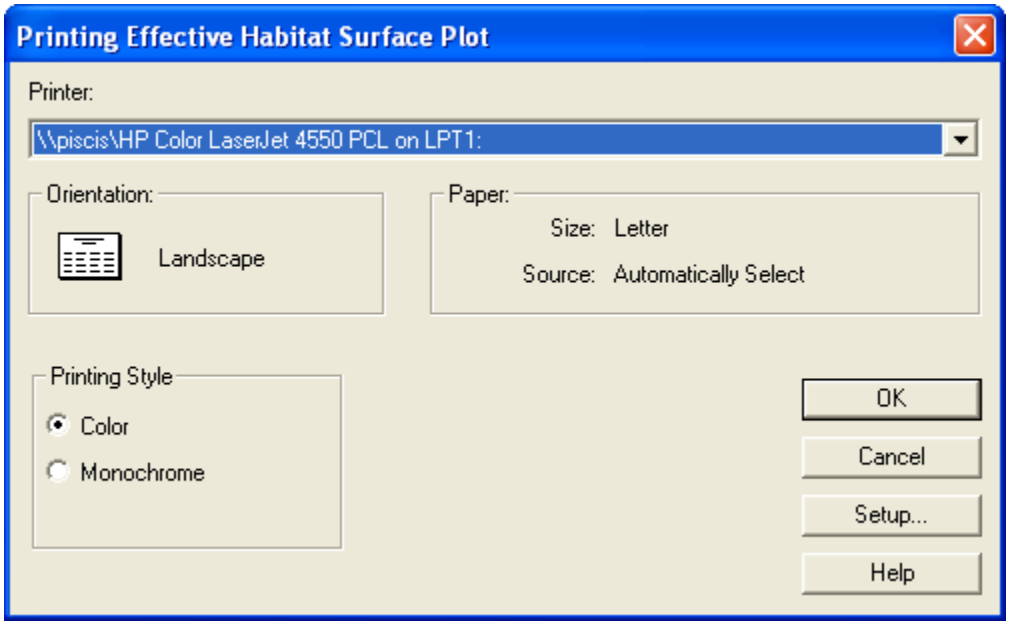
## Parts of the 1D Habitat Viewer Plug-in

### *Graph Window*

The graph window displays the graphs selected using the graph tabs.

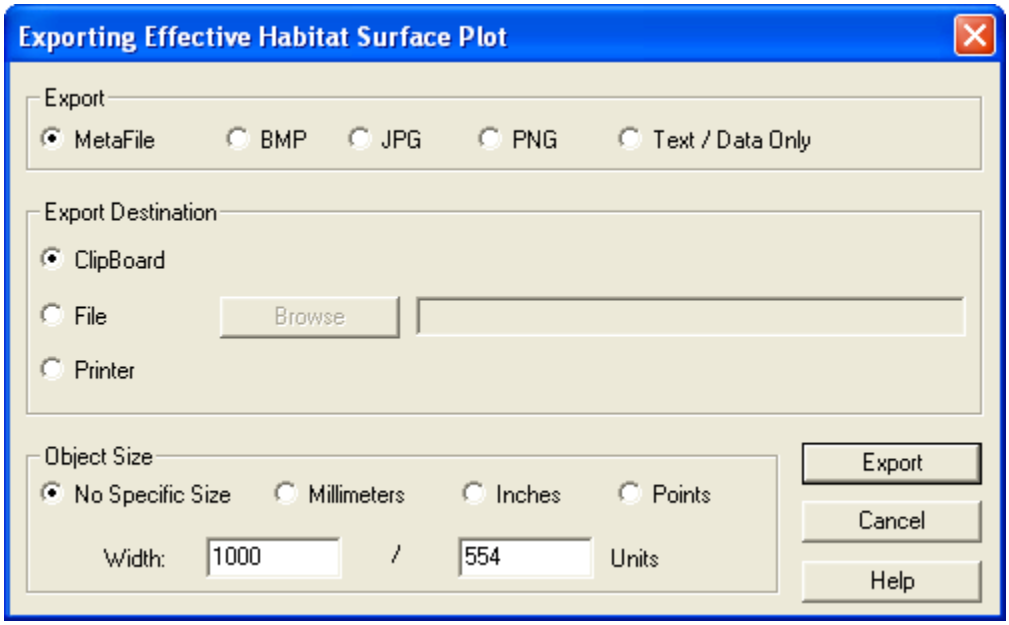
### *Print Button*

Use this button to print the current graph in the graph window. The print dialog appears as shown in the following figure. Options include printer selection, paper layout, and printing style (color, monochrome or monochrome with symbols).

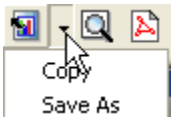



### *Export Button*

The export button allows a user to export the current graph to the clipboard or to a file in one of several formats. When the export button is clicked, the following form appears:



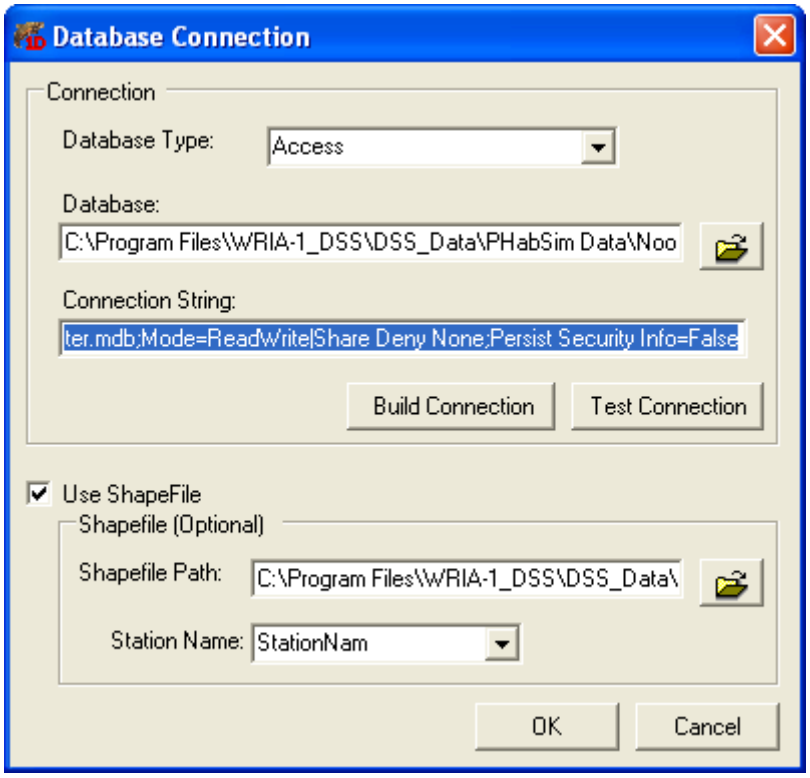

"Metafile" format is best for publication quality reports. "JPG" or "PNG" are best for use on web sites. If the data underlying the plot is needed, then use the "Text / Data Only" option.

Clicking on the arrow next to the Export button 

will give the user the option to copy or save the currently displayed graph.

***Database Connection Button***

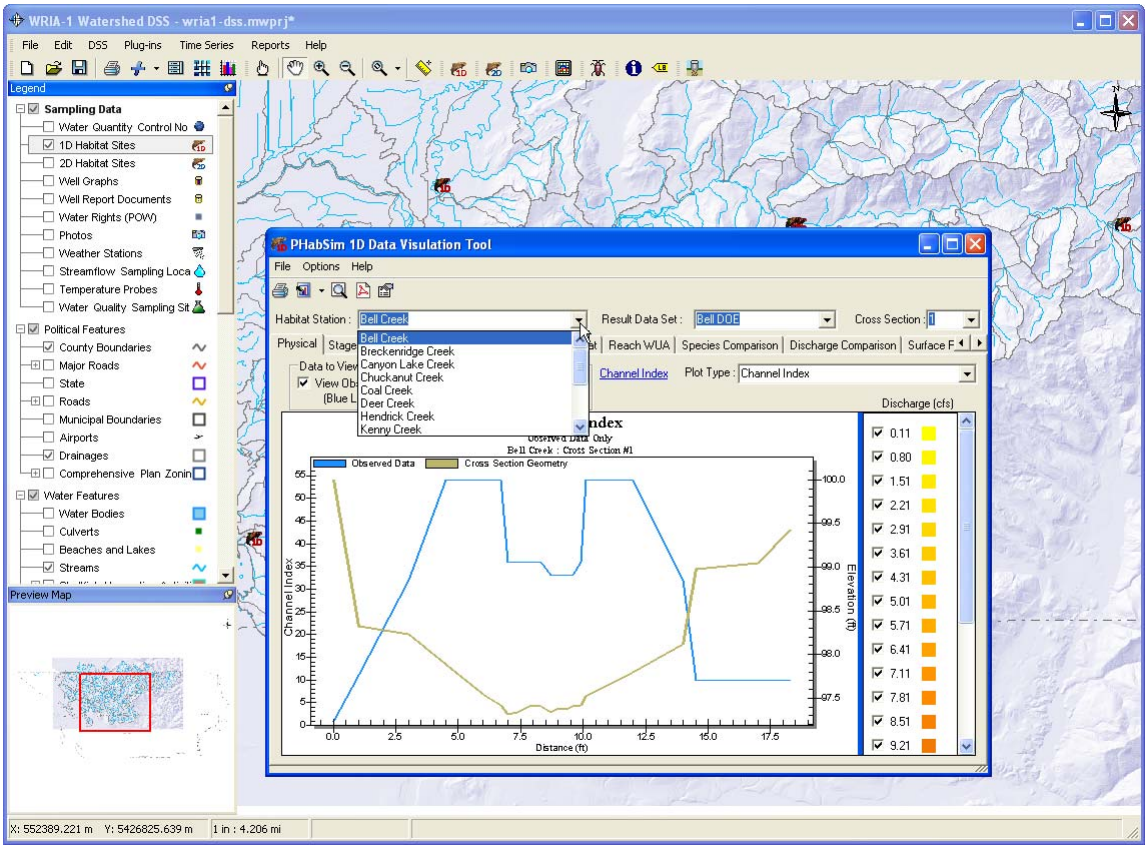
This button is used to display the database connection information for this plug-in. When the button is clicked, the following form is displayed:




Users will typically not need to modify any of the settings in this form. However, if a different PHABSIM database or shape file were to be used then this form is where the new database connection information would be assigned.

***Habitat Station Selector***

The station selector button is used to select different habitat stations where 1-D PHABSIM results data sets are available. The stations in this drop-down list correspond to stations shown in the main DSS map in the "1D Habitat Sites" layer as shown in the following figure:




### *Result Data Set Selector*

The data set selector is used to select specific data sets at each habitat station. Often there is only one data set for each station. However, if more than one data set is available at the station, they will all appear in the data set selector.

### *Cross Section Selector*

The cross section selector is used to select specific cross sections that are part of a data set at a particular habitat station. Cross section data are collected at one or more cross sections at the habitat station. Available cross sections can be selected using this selector. The physical cross section geometry for the selected cross section is shown on the first graph tab titled “Physical”. Each of the graph tabs is described in the following sections

### *Other Features*

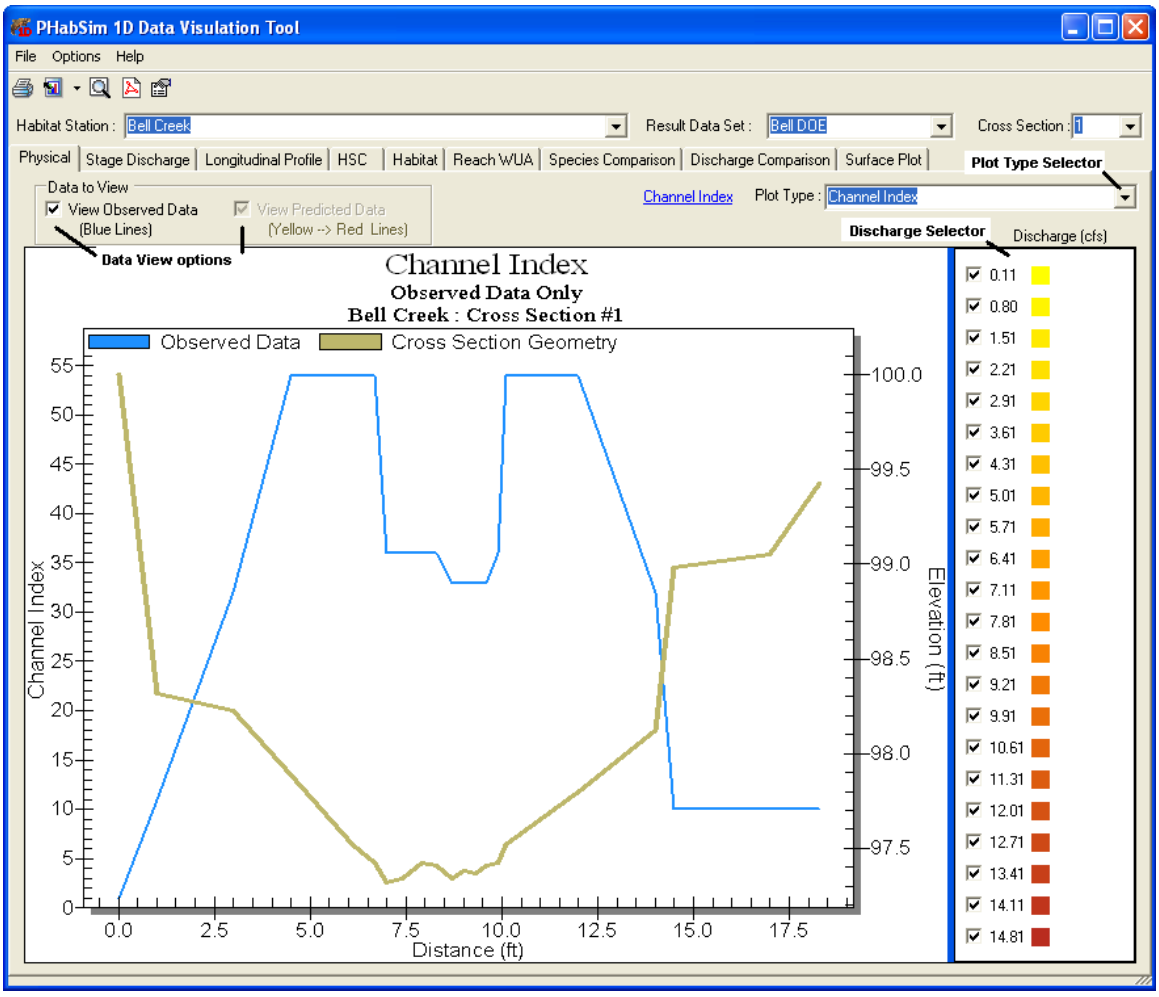
There are also Menu Options available. Under the File menu you can Copy, Save, or Print the current graph, or you can Exit the Viewer. These exporting functions can also be accessed through the Export Button. Under the Help menu you can launch and view the User’s Manual (this document). Finally, under the Options menu you can launch the Database Connection form, the same as pressing the Database Connection Button.

# PHabSim 1D Data Visualization Tool Graphs

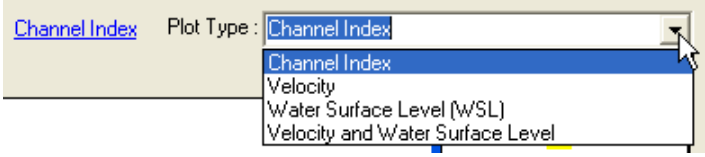
The PHabSim 1D Data Visualization Tool includes nine tabs allowing the user to view various plots of the 1D data. Each of these tabs is descried in detail in the following sections.

## *Physical*

The Physical tab shows a scaled view of the selected cross-section geometry. The X-axis is the distance across the cross section in feet. The Y-axis on the right side of the graph is the elevation of the bed in feet. The Y-axis on the left hand side of the graph displays the units and scale for the Plot Type selected using the Plot Type Selector box shown below. The Plot Type selected for this graph is Channel Index.

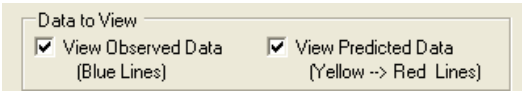



By clicking on the Plot Type Selector menu, the user can select to display the Channel Index associated with the selected cross section, the velocity sets, water surface levels or both the velocity sets and water surface levels as shown below.




Notice that when the Channel Index plot type is selected, a blue underlined Channel Index link is visible to the left of the Plot Type window. Clicking on this link will bring up an explanation of the channel index codes used for this project.

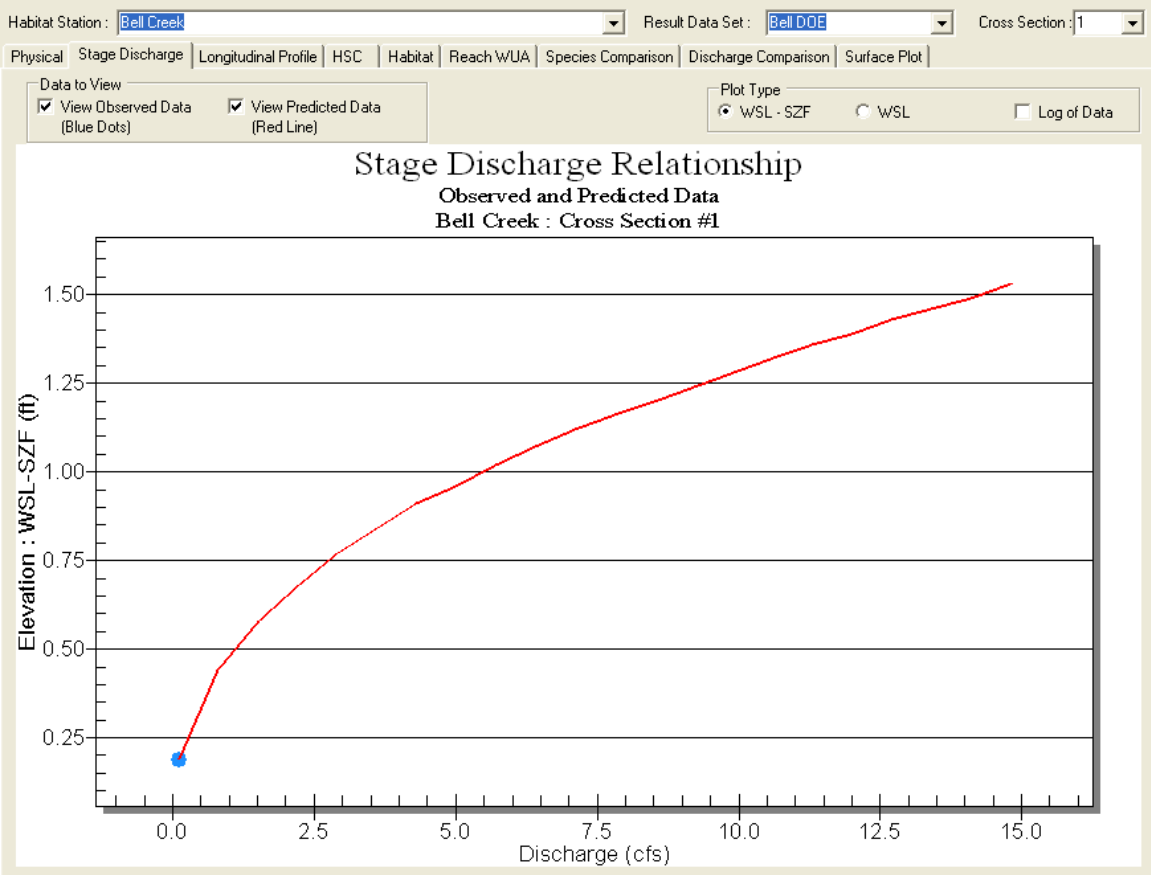
Along with these Plot Type options, the user also has the ability to display or hide the observed data and predicted data values. This can be done by checking and unchecking the boxes next to these data view options. Note that Channel Index has only observed data, thus the predicated data option is not available. For the velocity and water surface level plot options, the observed and predicted values can be displayed using these checkboxes as shown below.




The Y-axis along the left hand side of the graph will change to show velocity in feet per seconds and water surface levels in feet. When displaying the velocity and/or water surface level plot options, the user may also select which of the predicted discharges (or flows) to display results for using the Discharge Selector window. All the predicted discharges used in the PHabSim model runs will be displayed in this window located on the right hand side of the graph window. By using the check boxes next to each discharge (listed in cfs), the user may choose which discharges to display. This option will change the plotted results only if the View Predicted Data checkbox is selected.

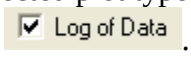
### *Stage Discharge*

The Stage Discharge tab plots the relationship between the observed and predicted stage discharges. The X-axis is the Flow value for the stage discharge in cubic feet per second (cfs). The Y-axis is the Stage Discharge value in feet. The Y-axis values are dependent on the plot type selected in the “Plot Type” section.




The Observed Data points are shown in blue. The x-values are the actual flows observed in the Calibration Data set. The Predicted Data is drawn in red. The x-values are the predicted flows that are available.

The user can toggle on and off the Observed Data and the Predicted Data by checking and un-checking the checkboxes marked “View Observed Data” and “View Predicted Data” in the “Data to View” section.

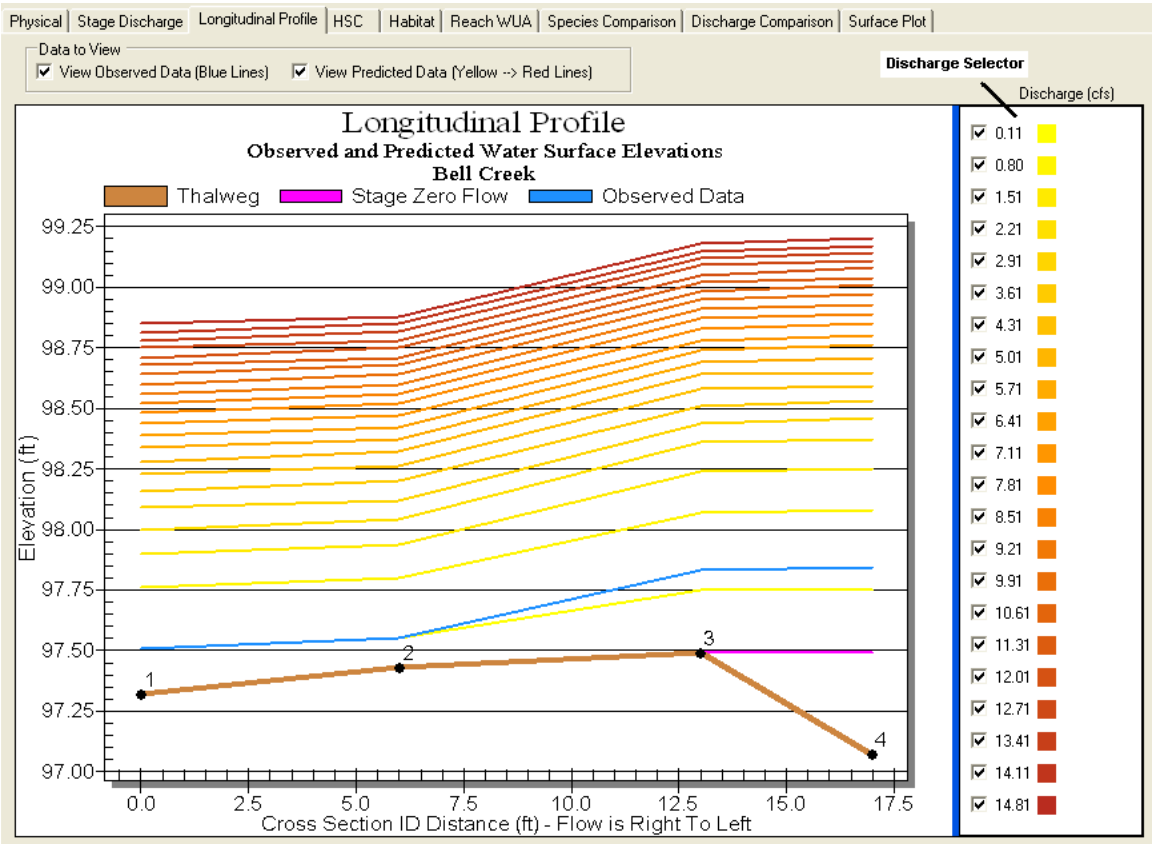
There are two Plot Types available: Water Surface Level minus the Stage Zero Flow value (“WSL – SZF”), or just the Water Surface Level (WSL). Either of these plot types may also be viewed on a Log scale. To view the selected plot type in a Log scale simply check the checkbox next to the Log of Data option. 

.

To view the selected plot type in a linear scale, simply un-check the checkbox. The default is to have the “Log of Data” checkbox unchecked. This means that the data will be displayed in a linear scale.

### *Longitudinal Profile*

The Longitudinal Profile tab plots the longitudinal profile for the observed and predicted data sets. The X-axis shows the distance between the Cross Sections where cross section 1 starts at 0 feet. The Y-axis is the Elevation in feet. The user may specify which discharges they wish to display using the Discharge Selector box along the right hand side of the window and is labeled in the example below. There is a check box for each of the Predicted Flows.




The Thalweg is the brown line and the Stage at Zero Flow (szf) is the magenta line, as shown in the example above and indicated in the Legend at the top of the plot. The Observed data are the blue lines. There is a line for each of the Calibration data sets collected.

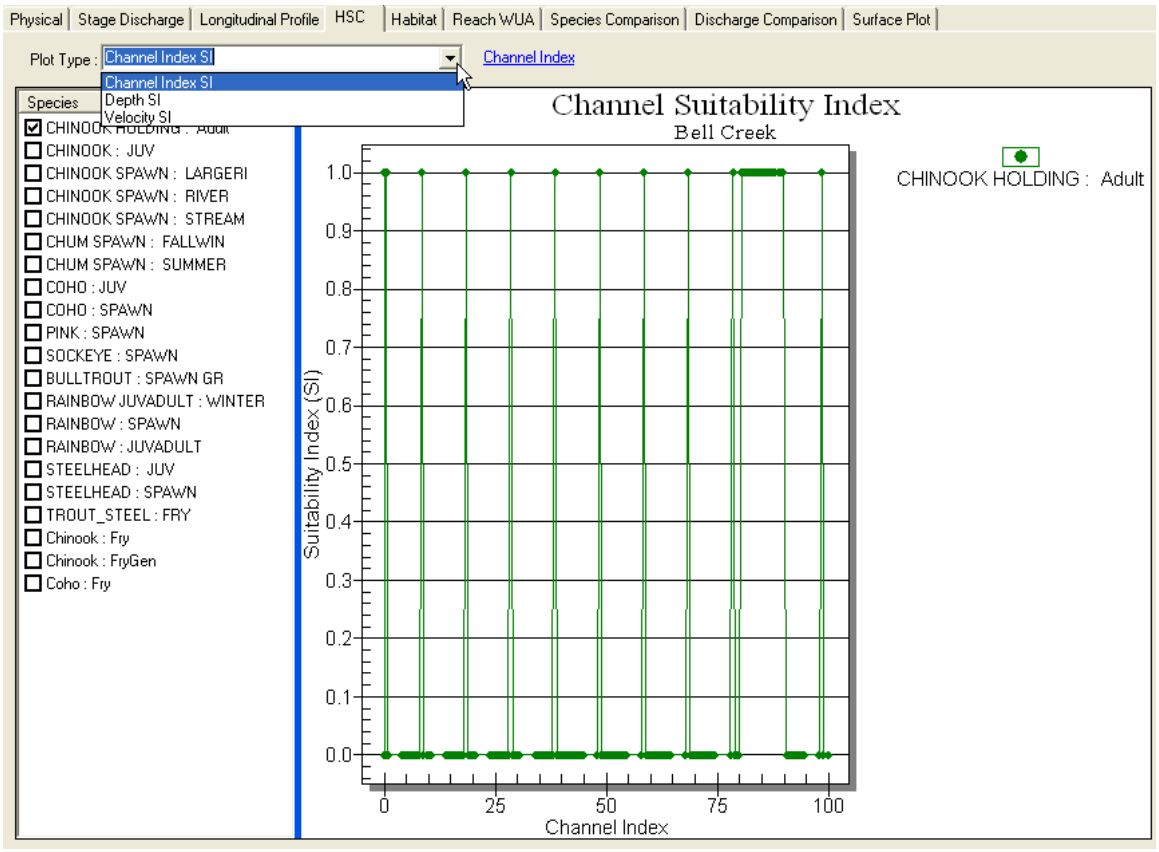
To view the Observed Data, the checkbox marked “View Observed Data” must be checked. To view the Predicted Data, the checkbox marked “View Predicted Data” must be checked. If the Predicted Data is checked then to have the line drawn for a certain Predicted Flow, the checkbox for it must be checked. To turn off the line, simply uncheck the checkbox in the Discharge Selector window. The default is for all of the checkboxes, both in the “Data to View” section and for each of the Predicted Flows, to be checked. This means that a line for each of the Predicted Flows and all of the Observed

Data will be drawn. The plot is automatically updated when a checkbox is checked or unchecked.

Note that since this is a Longitudinal Profile it is done across all of the cross sections so the Cross Section selector option is not available.

### *HSC*

The HSC tab displays the Habitat Suitability Criteria used to model each species and lifestage. The Plot Type selector allows the user to select a plot of the Channel Index suitability index, depth suitability index, or the velocity suitably index by selecting the desired option from the pull down menu as shown below. Notice the blue underlined link called Channel Index is available to the right of the Plot Type menu when the channel index option is selected. Clicking on this link displays a description of the channel index codes.



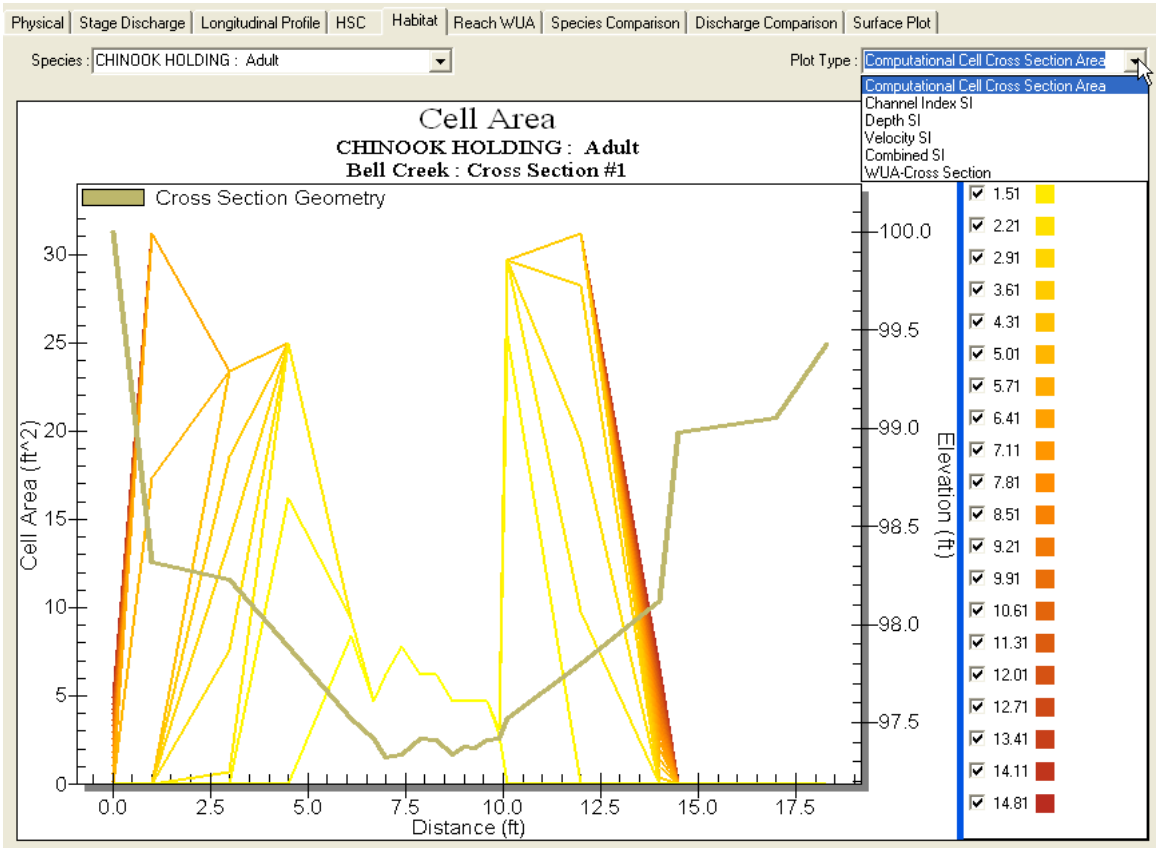

The Y-axis for all plot types is the Suitability Index (SI) on a scale from 0.0 (completely unsuitable) to 1.0 (suitable). The X-axes depend on the plot type selected. For Channel Index, the X-axis displays the Channel Index codes. Again, a description of these codes can be found by clicking on the blue Channel Index link to the right of the Plot Type selector menu. The X-axis for the Depth SI is the depth in feet, and the Velocity SI has units of feet per second (ft/s).

The user may also select multiple species and lifestages to be displayed by checking and unchecking the boxes next to the species and lifestage names in the Species window located along the left hand side of the graph window.

All of the Suitability Index values apply to every cross section at the selected site, so the cross section selector option is not available for this tab.

### *Habitat*

The Habitat tab displays habitat related data for the species selected from the Species dropdown combo-box for each of the Predicted Flows. The X-axis is the distance along the cross section in feet. The Y-axis on the left side of the plot is dependent on the plot type selected from the Plot Type dropdown combo-box. For example, if the selected plot type is Computational Cell Cross Section Area (as seen below) then the Y-axis values would be the area for the given point along the cross section in square feet (ft^2). The units for the Y-axis on the right side of the plot are for the cross section geometry elevation given in feet.

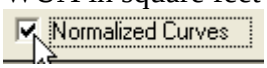



In the example shown above, the tan line is the cross-sectional profile. The other colored lines correspond to the colors in the boxes next to the checkbox for each Predicted Flow.

On the right-hand side are checkboxes for each of the Predicted Flows. To have the line drawn for a certain Predicted Flow, the checkbox for it must be checked. To turn off the line, simply uncheck the checkbox. The default is for all of the checkboxes to be checked, this means that a line for each of the Predicted Flows will be drawn. The plot is automatically updated when a checkbox is checked or unchecked.

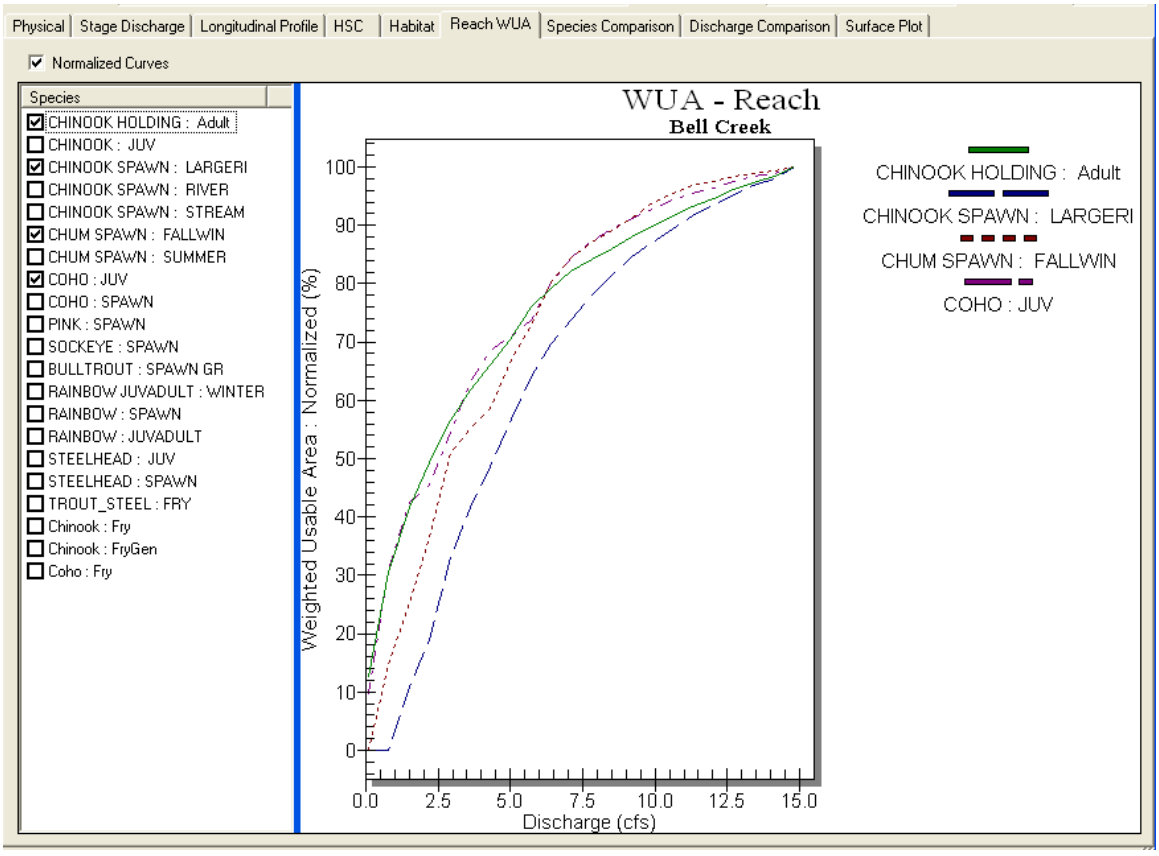
### *Reach WUA*

The Reach WUA tab displays the weighted usable area (WUA) for all predicted flows at the selected reach. The X-axis displays the discharge in cfs and the Y-axis shows the WUA in square feet per 1000 feet of reach (ft^2/100ft). By checking the




option located above the Species list on the left hand side of the window, the user can view the plots normalized as a percent of the maximum habitat value.

This option is useful when comparing several species, which can be done by checking and unchecking the boxes next to the species and lifestages listed in the Species window. The example below shows the Normalized Curves for four species/lifestages. The legend to the right of the plot lists which species are selected.




As many as 12 species/lifestages can be displayed simultaneously using this tool.

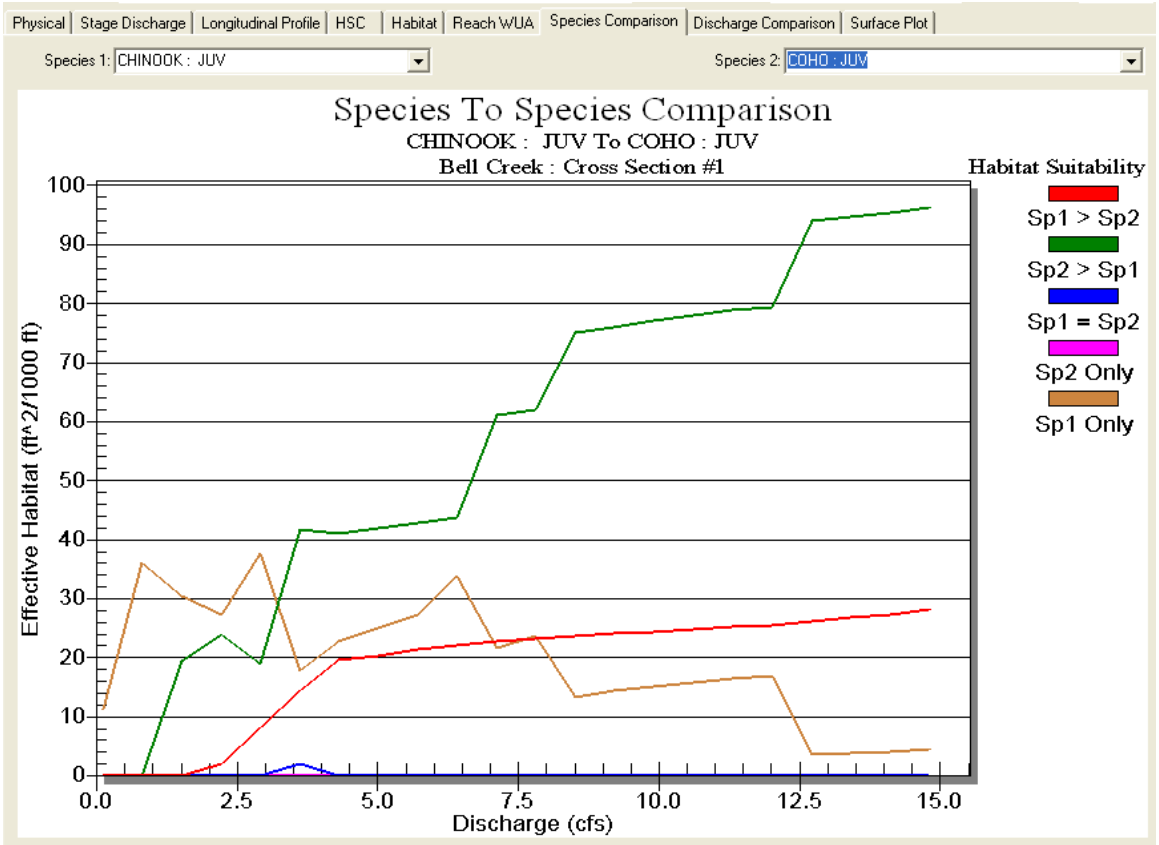
### *Species Comparison*

The Species Comparison plots the Effective Habitat for the comparison of two species. To select the species to compare, select the first species from the Species 1 dropdown combo-box. Then select the second species for comparison from the Species 2 dropdown combo-box.

The X-axis values are the Predicted Flow values in cubic feet per second (cfs). The Y-axis is the Effective Habitat values for the given predicted flow. The line colors and meanings are shown in the Legend on the right of the graph.

The red line represents where the habitat suitability for Species 1 is greater than the habitat suitability of Species 2. The green line shows where the habitat suitability for Species 2 is greater than the habitat suitability of Species 1. The blue line shows where the habitat suitability for both species is equal. The magenta line shows the habitat suitability for Species 2 only, and the tan line shows the habitat suitability for Species 1 only.

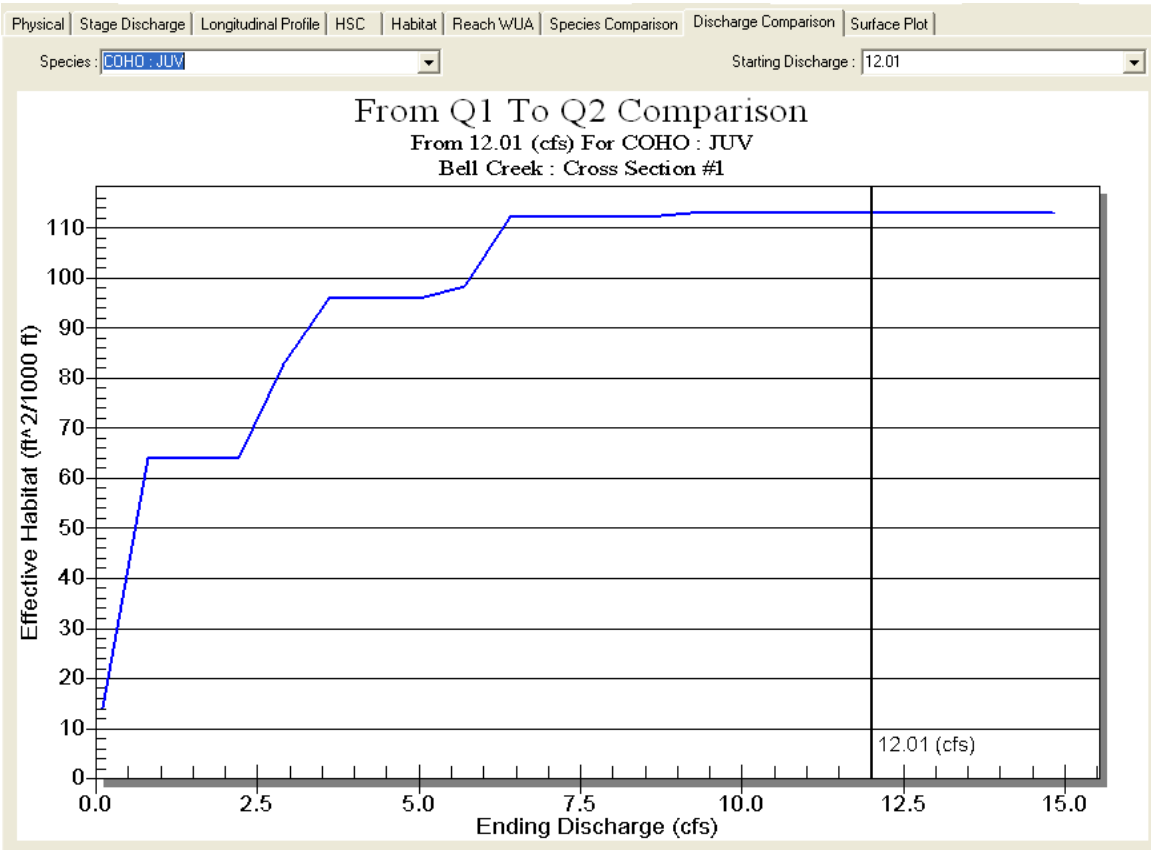
Note that if the same species is selected for Species 1 and Species 2, only the blue line (equal) will appear. The example below shows Chinook Juvenile compared to Coho Juvenile at cross section 1 at Bell Creek.



### *Discharge Comparison*

The Discharge Comparison tab plots the Effective habitat for the selected species starting at the predicted discharge (flow) value is changed to the selected "Starting Discharge" value. The black vertical line represents the selected flow. The available flows for comparison are the same as the Predicted Flows.

The X-axis shows the entire range of Predicted Flows in cfs. The Y-axis value is the Effective Habitat value in square feet per 100 feet (ft^2/100ft). The example below shows the entire effective habitat curve since the starting flow is so near the end flow value. As the starting flow value is decreased, the chart will automatically update itself to show only that portion of the curve.



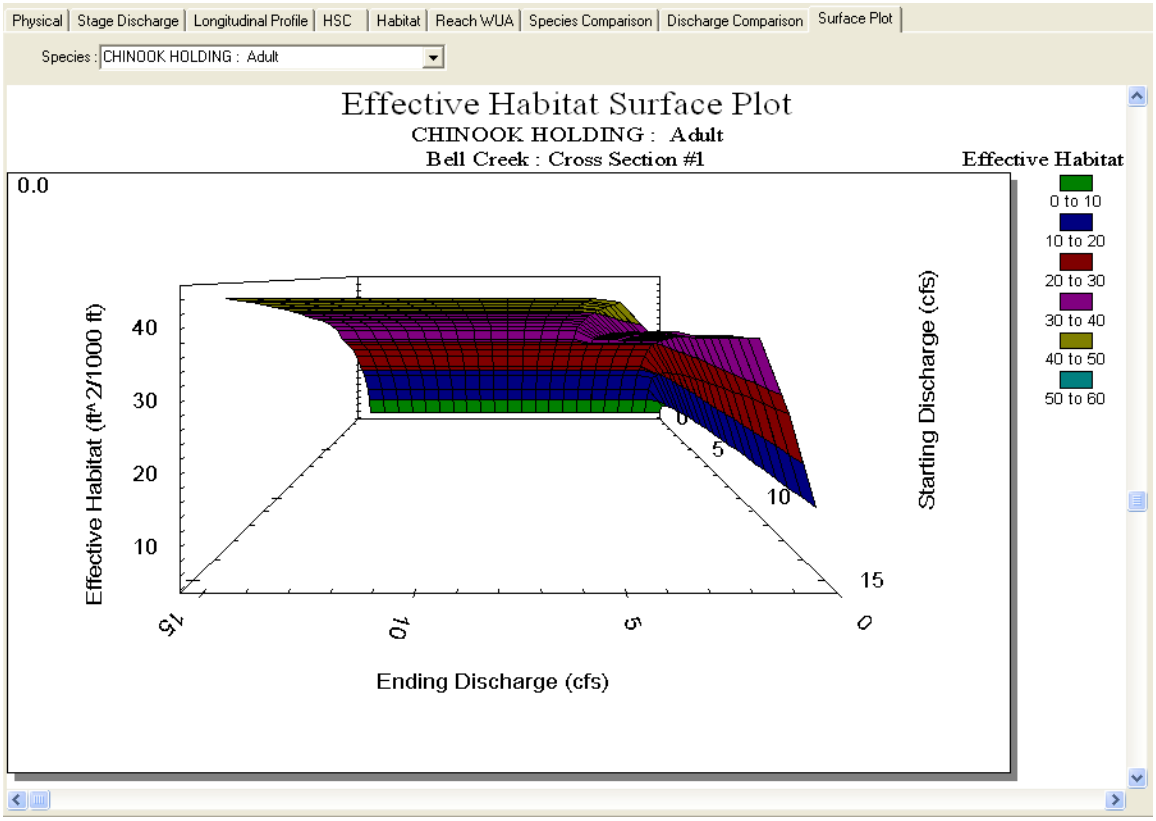

### *Surface Plot*

The Surface Plot is a 3-Dimensional wire-frame visualization of all of the available flows in the Flow Comparison tab stacked up next to each other for the selected species from the Species dropdown combo-box. The X-axis and Z-axis correspond to the available Predicted Flows in cubic feet per second (cfs). The Y-axis is the Effective Habitat value.

There are two methods for rotating the view of the graph. The first method is to double click the plot. This will cause the graph to begin to rotate. It will continue to do so until it is double clicked again, at which point it will stop. The number in the top-left corner of the plot indicates the current degree of rotation.

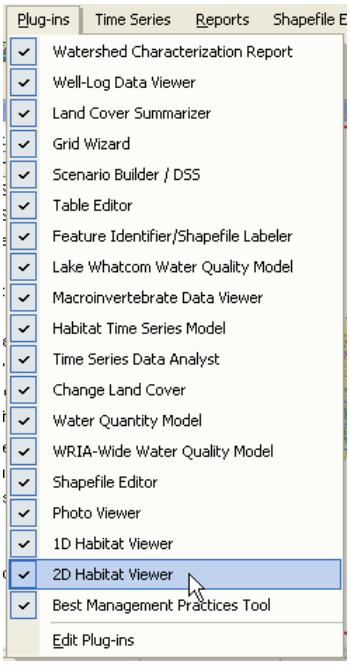
The second method is to use the scroll bars located at the bottom of the plot. When the scrollbar is at its left-most position, the degree of rotation is 0.0. As you move it to the right, it rotates the picture. The degree of rotation is updated in the top-left corner of the plot. When the scrollbar is at its right-most position, the degree of rotation is 359.0. The default position for the scrollbar is the left-most position or 0.0 degrees of rotation.

The scrollbar, located on the right-hand side of the plot, is to change the perspective of the graph. As the scrollbar is moved upward, the user will see the plot from the underside of the wire-frame (looking up from the bottom on the plot). If the scrollbar is at its top-most position, the user will see the "underside" of the wire-frame plot. As the scrollbar is moved downward, the user will see the plot from the top of the wire-frame (looking down on the plot). If the scrollbar is at its bottom-most position, the user will see the "top-side" of the wire-frame plot.

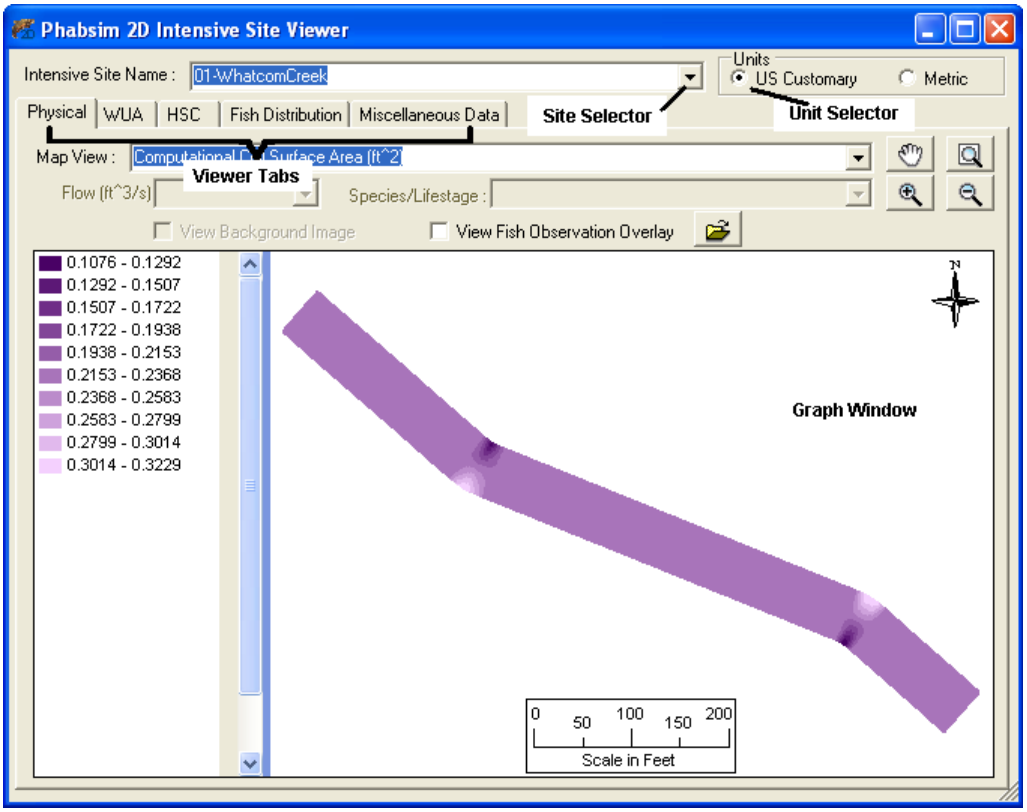


# 2D Habitat Viewer Plug-in

The 2D Habitat Viewer plug-in allows you to view and query information on the effects to fish habitat of different stream flow regimes. It supports 2D modeling output at all 2D intensive sites. This plug-in is launched by clicking on the Phabsim 2-D Intensive Viewer button on the toolbar, . If this button is not available on the toolbar, click on the Plug-ins menu on the menu bar and select 2D Habitat Viewer from the list as shown below.



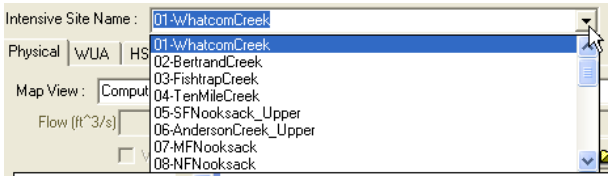

This will add the 2D Habitat Viewer button to the toolbar. When the toolbar button is clicked, the following form will appear. In the figure below, the different parts of the form are labeled for reference. The subsequent sections describe each part of the viewer.




# Parts of the 2D Habitat-Viewer Plug-In

### *Site Selector*

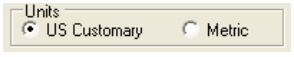
The Site Selector option allows the user to select which of the Intensive Sites to view. To choose a site, click on the arrow button on the right edge. This will open a list of the sites available for viewing as shown below.




This option is only available for Viewer Tabs where individual site data is available. Tabs containing data related to all sites will not allow this option.

### *Unit Selector*

The Units Selector allows the user to change between US Customary units and Metric units. Changing the units will automatically update the units displayed in the Graph Window.



### *Viewer Tabs*

The five Viewer Tabs allow the user to view the different results for the selected site. Each Viewer Tab has a unique Graph Window and displays unique data for the site selected. The following sections describe the Viewer Tabs in detail.

### *Graph Window*

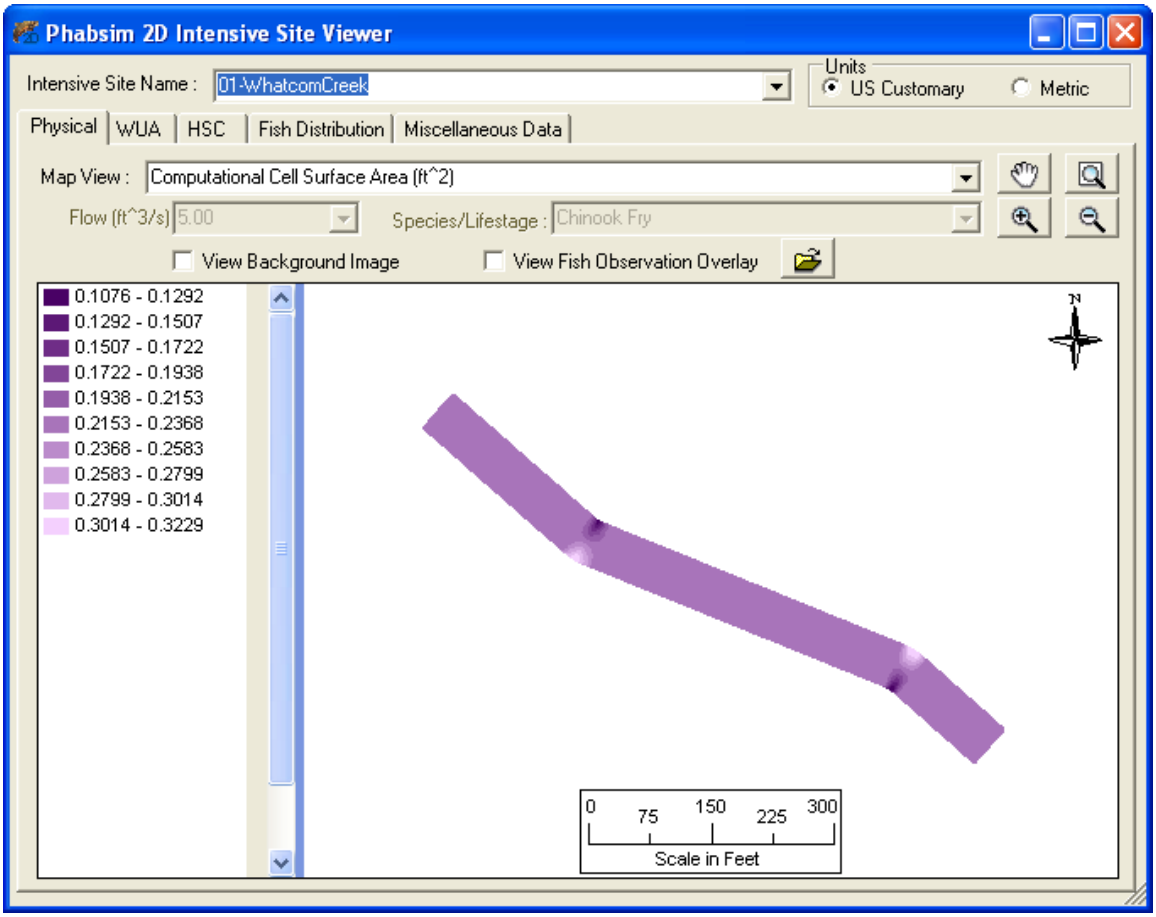
The graph window displays the selected options for the site selected. This window will vary depending on the options selected and the Viewer Tab selected.

## 2D Data Viewer Graph Tabs

The 2D Data Viewer includes five tabs allowing the user to view various pieces of the 2D data. Each of these tabs is descried in detail in the following sections.

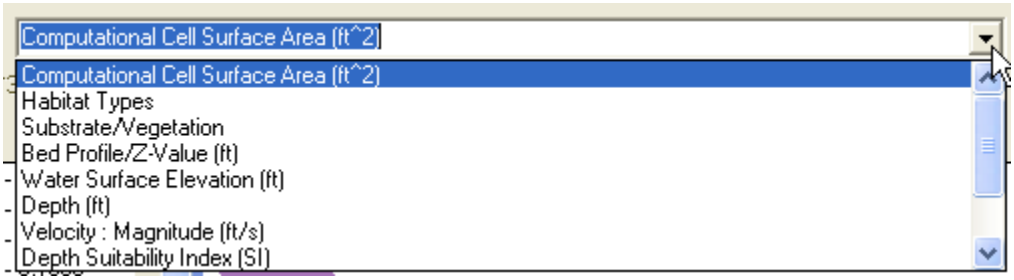
### *Physical*

The Physical tab displays the mesh properties for the selected site, as well as background images and fish observations. When the 2D Viewer first opens, the screen will appear as shown below with the Computational Cell Surface Area for Whatcom Creek showing.

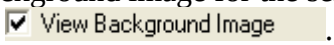



By selecting the pan button, the user may pan around the image displayed in the graph window. The user may also zoom in and zoom out or zoom to the full extent of the image .

Using the Map View selector, the user may choose what information is displayed in the Graph Window.



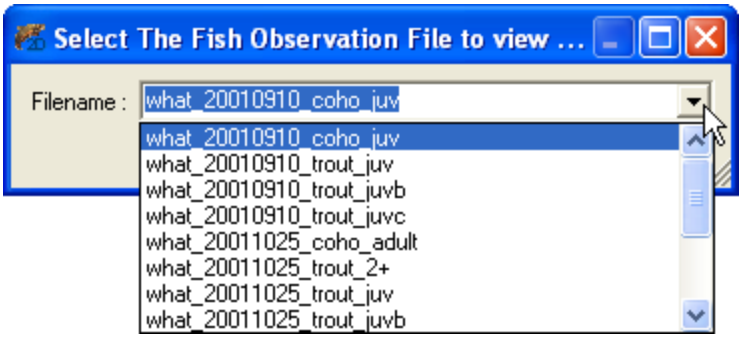

The units for each option are displayed in parenthesis after the option name. Once the map view option is selected, the graph window will be updated to display that information. A legend is displayed on the left side of the graph window.

The user may also display the background image for the selected site by checking the box next to View Background Image 

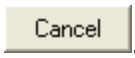
. Once the image loads, it will be displayed underneath the map view data.

Check the box next to View Fish Observation Overlay

to view any fish observation overlays at the selected site. Once this box is checked, a window will pop-up allowing the user to choose which of the available fish observations to view. Use the arrow to the left of the filename to access the entire list of fish observation files as shown below.

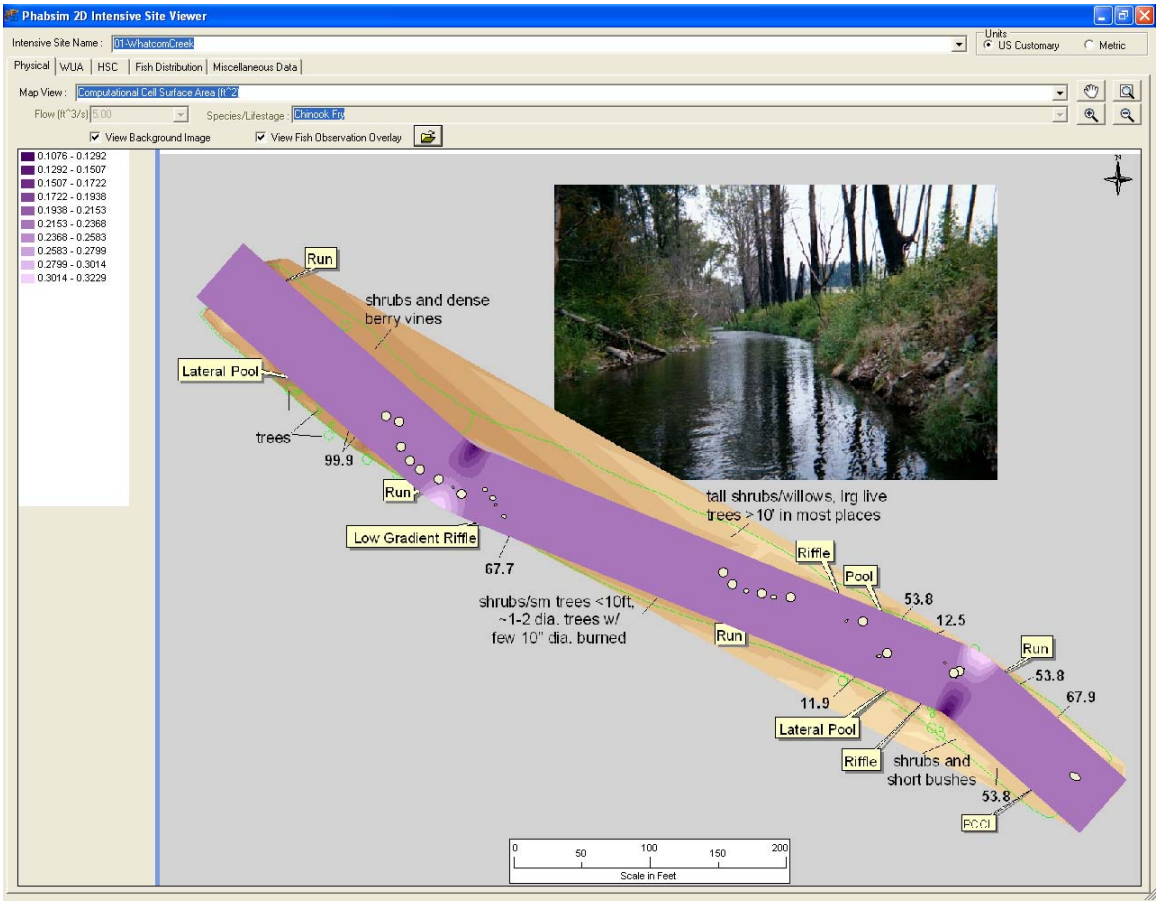



Once the desired fish observation has been selected, press

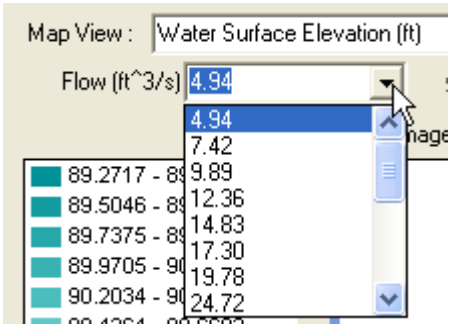
to add it to the graph window. To exit the Select Fish Observation window without adding anything to the graph window, press 

.

Adding fish observations and background images may take a few seconds. To select a different fish observation file to view, press the button to the right of the View Fish Observation Overlay option. This will bring up the Select Fish Observation window again and allow the user to select another file for viewing. Only one fish observation can be viewed at a time. The example below shows the Whatcom Creek computational cell surface areas with background image and steelhead juvenile fish observation overlay.




Some of the map view options, such as Water Surface Elevation, Depth, and Velocity Magnitude, allow the user to view data at different flows. When this is an option, the Flow Selector option will be activated and the user can select flows using the arrow to the left of the Flow Selector as shown below.




Once a flow is selected, the graph window will be updated to show the selected data. Other options, such as Depth Suitability Index, Velocity Suitability Index, Substrate/Vegetation Suitability Index, and Combined Suitability Index, allow the user to view data at different flows and for different species. The flows are selected in the same manner as shown above. To select a certain species and/or lifestage, the user can select the arrow button to the right of the Species/Lifestage Selector option as shown below.

Once the desired species/lifestage has been selected, the graph window will be updated to display that information.

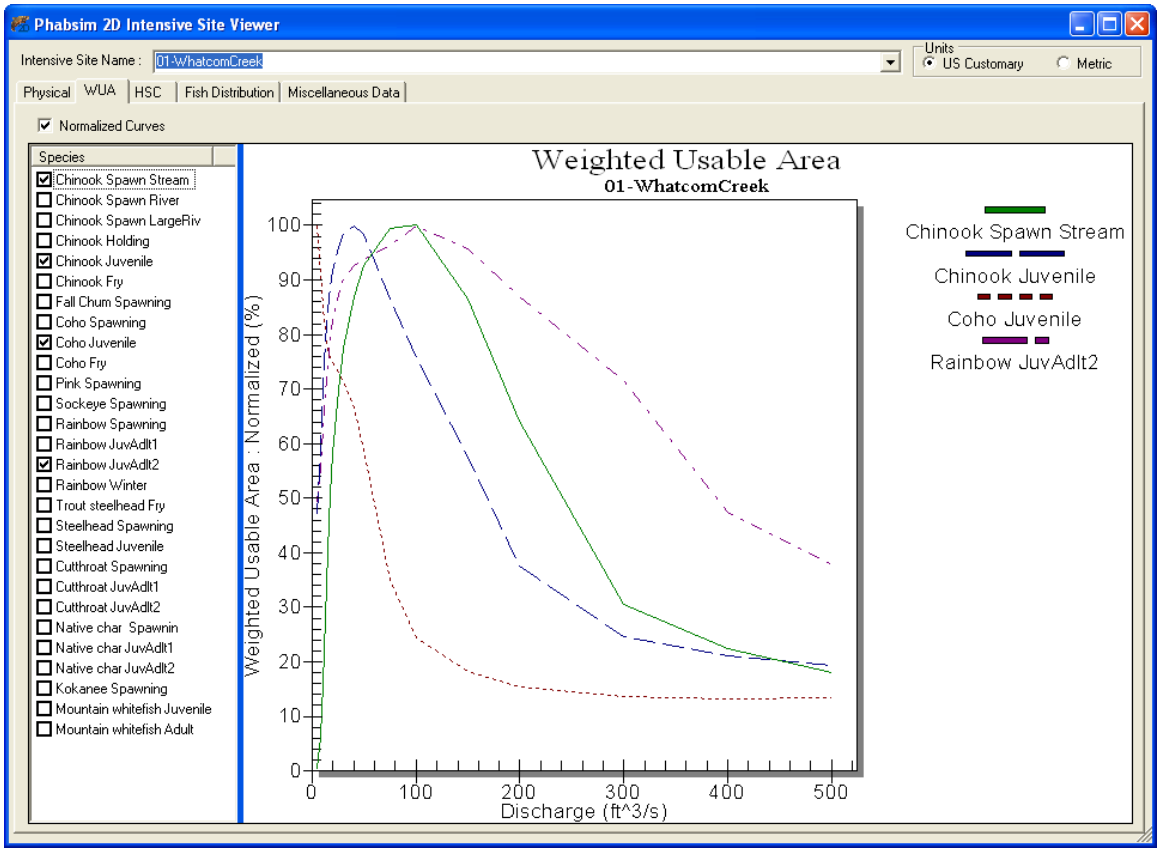
***WUA***

The WUA tab displays the weighted usable area (WUA) for all predicted flows at the selected reach. The X-axis displays the discharge in cfs and the Y-axis shows the WUA in square feet per 1000 feet of reach (ft^2/100ft) if US Customary is selected or square meters per 100 meters (m^2/100m) if Metric is selected for the units option.

By checking the

option located above the Species list on the left hand side of the window, the user can view the plots normalized as a percent of the maximum habitat value.

This option is useful when comparing several species, which can be done by checking and unchecking the boxes next to the species and lifestages listed in the Species window. The example below shows the Normalized Curves for four species/lifestages. The legend to the right of the plot lists which species are selected.

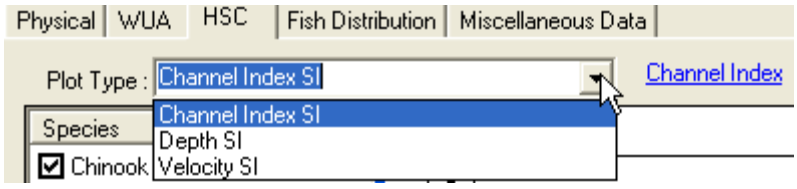



As many as 12 species/lifestages can be displayed simultaneously using this tool.

***HSC***

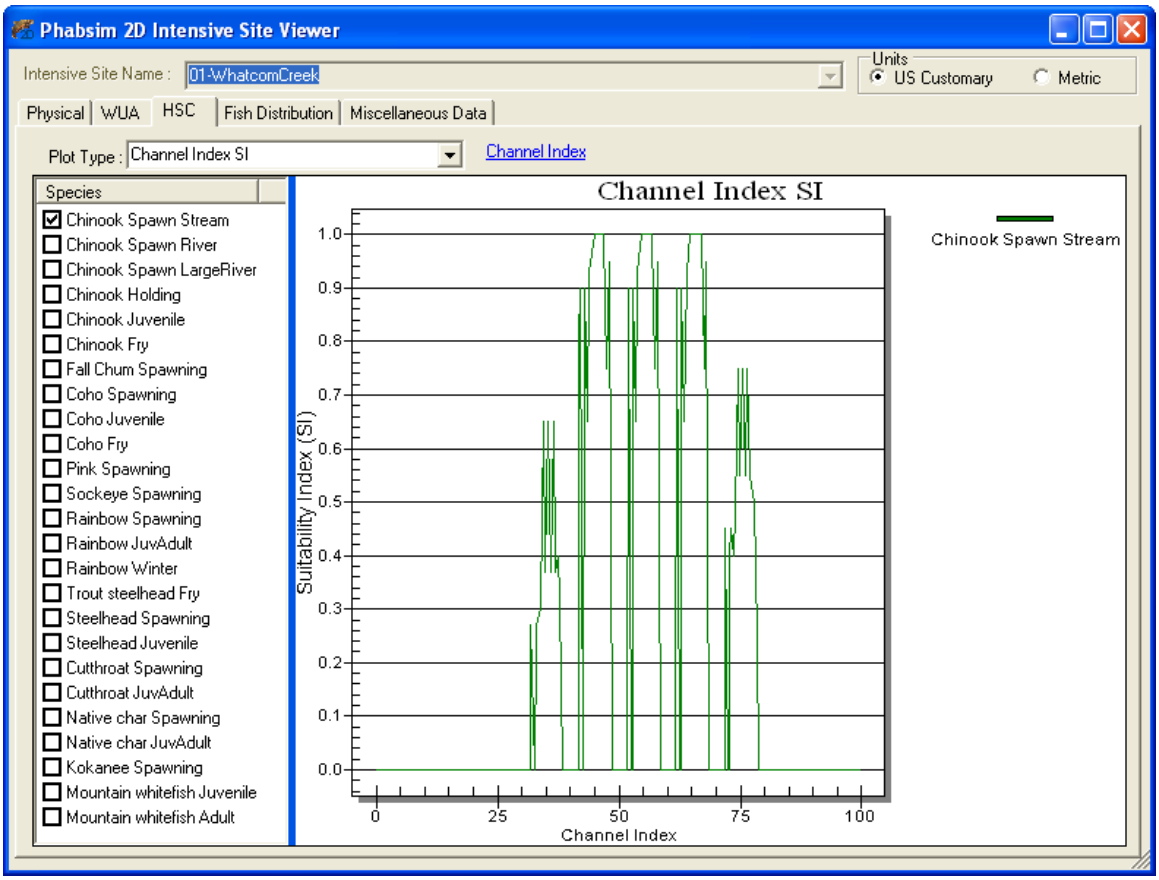
The HSC tab displays the Habitat Suitability Criteria used to model each species and lifestage. This information is used for all the sites, so the Site selector option is not available.

The Plot Type selector allows the user to select a plot of the Channel Index suitability index, depth suitability index, or the velocity suitably index by selecting the desired option from the pull down menu as shown below.



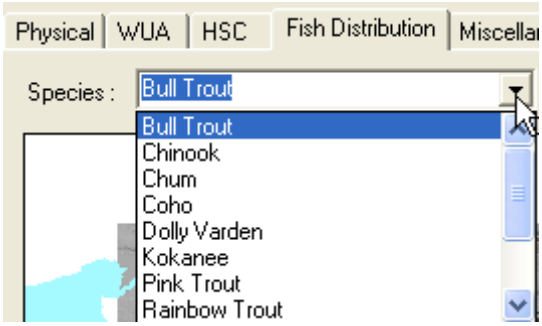

Notice the blue underlined link called Channel Index is available to the right of the Plot Type menu when the channel index option is selected. Clicking on this link displays a description of the channel index codes.

The Y-axis for all plot types is the Suitability Index (SI) on a scale from 0.0 (completely unsuitable) to 1.0 (suitable). The X-axes depend on the plot type selected. For Channel Index, the X-axis displays the Channel Index codes. Again, a description of these codes can be found by clicking on the blue Channel Index link to the right of the Plot Type selector menu. The X-axis for the Depth SI is the depth in feet or meters, and the Velocity SI has units of feet per second (ft/s) or meters per second (m/s) depending on which units are selected.

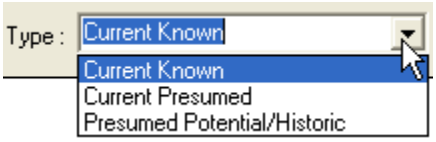



### *Fish Distribution*

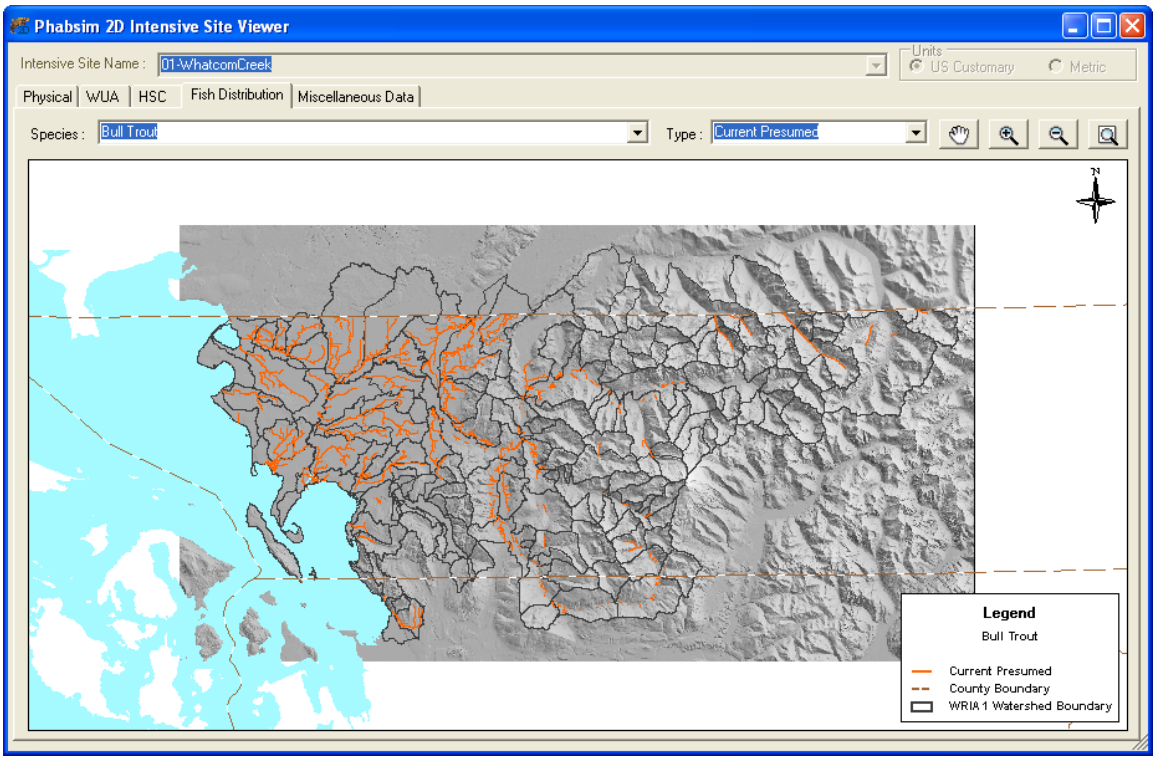
The Fish Distribution tab allows the user to view the fish distributions for certain fish species throughout the entire WRIA-1 watershed. Since this data is for the entire watershed, the Site selector option is not available. To select a species to display, click on the arrow to the right of the Species selector as shown below.




Once the desired species has been selected the graph window will update to display the selected information. The user may also select the type of distribution to display by clicking on the arrow to the right of the Type selector as shown below.




Once a distribution type is selected, the graph window will update to show the selected data. A legend showing the selected species and distribution type is shown in the bottom right corner of the graph window.

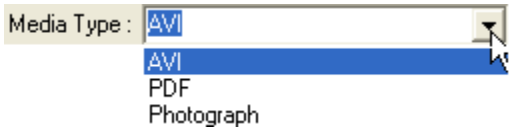


By selecting the pan button, located to the right of the Type selector, the user may pan around the image displayed in the graph window. The user may also zoom in and zoom out or zoom to the full extent of the image

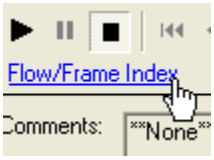
***Miscellaneous Data***

The Miscellaneous Data tab allows the user to view AVIs showing a movie of the combined suitability for the selected species/lifestages at the selected site, as well as photographs taken at the selected site, and PDF documents detailing the work at each site.

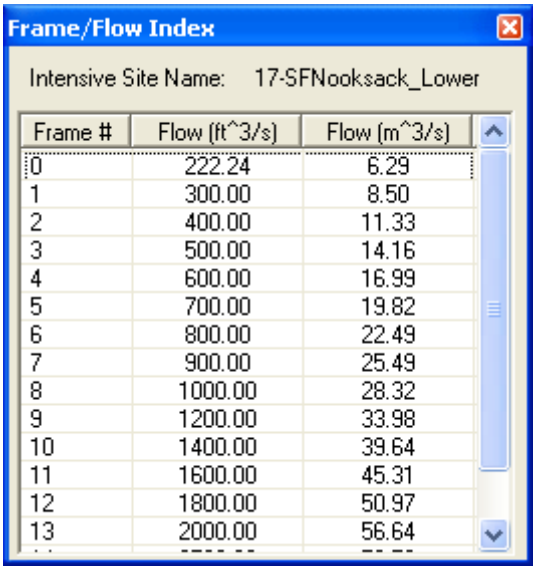
To select which type of data to view, use the arrow to the right of the Media Type selector as shown below.




AVIs show a background image for the study site, and often a photograph of a portion of the site. A legend is displayed in the top left hand corner of the screen. This shows the color scale used in the AVI to display the Combined Suitability. This scale ranges from unsuitable, a value of 0.000 as red, to highly suitable, a value of 1.000, as blue. A purple time clock located in the top right-hand corner of the window shows which frame is being displayed. For a list of flows associated with each frame, click the blue link located near the left hand bottom of the screen.



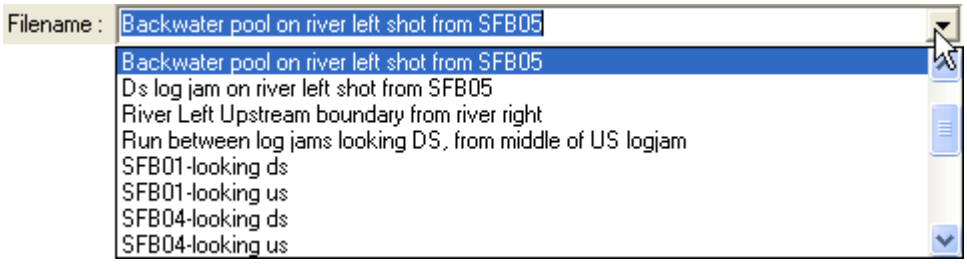

The Flow/Frame Index shows the frame associated with each flow in cfs and cms.




When the AVI is opened, it will automatically play. To stop or replay, use the buttons near the bottom left hand portion of the screen.

To view PDF files, Adobe Reader must be installed. PDF files are not available at all sites. To view a PDF file for the selected site, choose PDF from the Media Type selector as shown below. The PDF will open in Adobe Reader. If Adobe Reader is not installed, the PDF will not open. Adobe Reader can be downloaded free at www.adobe.com

Photographs taken at the selected site are available by selecting the Photograph option from the Media Types selector. Photographs are available at most sites, but not all. If more than one photograph is available at the selected site, the Filename selector window will become active. Clicking on the photograph filename will load the selected photo.

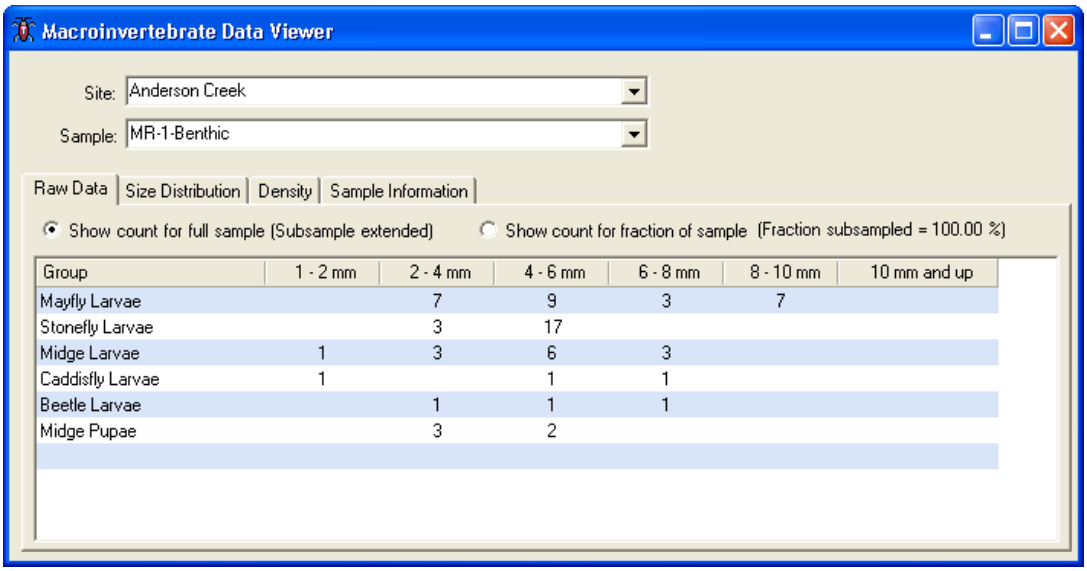

Filename :
Backwater pool on river left shot from SFB05
Backwater pool on river left shot from SFB05
Ds log jam on river left shot from SFB05
River Left Upstream boundary from river right
Run between log jams looking DS, from middle of US logjam
SFB01-looking ds
SFB01-looking us
SFB04-looking ds
SFB04-looking us

# Macroinvertebrate Data Viewer

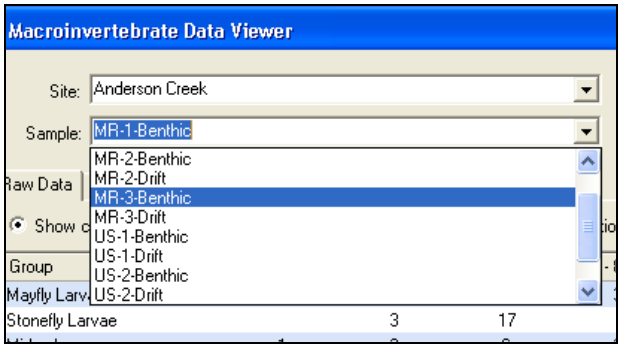
The Macroinvertebrate Data Viewer Plug-in to the DSS allows the user to see all of the macroinvertebrate data listed by site and sample. Data may only be viewed, not changed.

## Viewing Macroinvertebrate Data

Load the Macroinvertebrate Data Viewer plug-in by clicking on the Macroinvertebrate Data Viewer toolbar button . The Macroinvertebrate Data Viewer form will appear as shown.


Macroinvertebrate Data Viewer
Site: Anderson Creek
Sample: MR-1-Benthic
Raw Data
Size Distribution
Density
Sample Information
Show count for full sample (Subsample extended)
Show count for fraction of sample (Fraction subsampled = 100.00 %)
Group
1 - 2 mm
2 - 4 mm
4 - 6 mm
6 - 8 mm
8 - 10 mm
10 mm and up
Mayfly Larvae 7 9 3 7
Stonefly Larvae 3 17
Midge Larvae 1 3 6 3
Caddisfly Larvae 1 1 1
Beetle Larvae 1 1 1
Midge Pupae 3 2

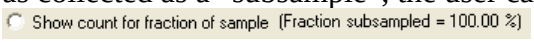
Click on the arrow on the right side of the Site window to reveal a drop down box listing all sites for which macroinvertebrate data was collected. Use the scroll bar to view the list and select a site by clicking on its name. Click on the arrow on the right side of the Sample window to see a list of all samples collected for the selected site. Scroll through the list to find the sample of interest and select it by clicking on its name.




## Macroinvertebrate Data Viewing Options

Tabs allow the user to view macroinvertebrate data in various formats. Options available in each format are discussed below.

### *Viewing Raw Data*

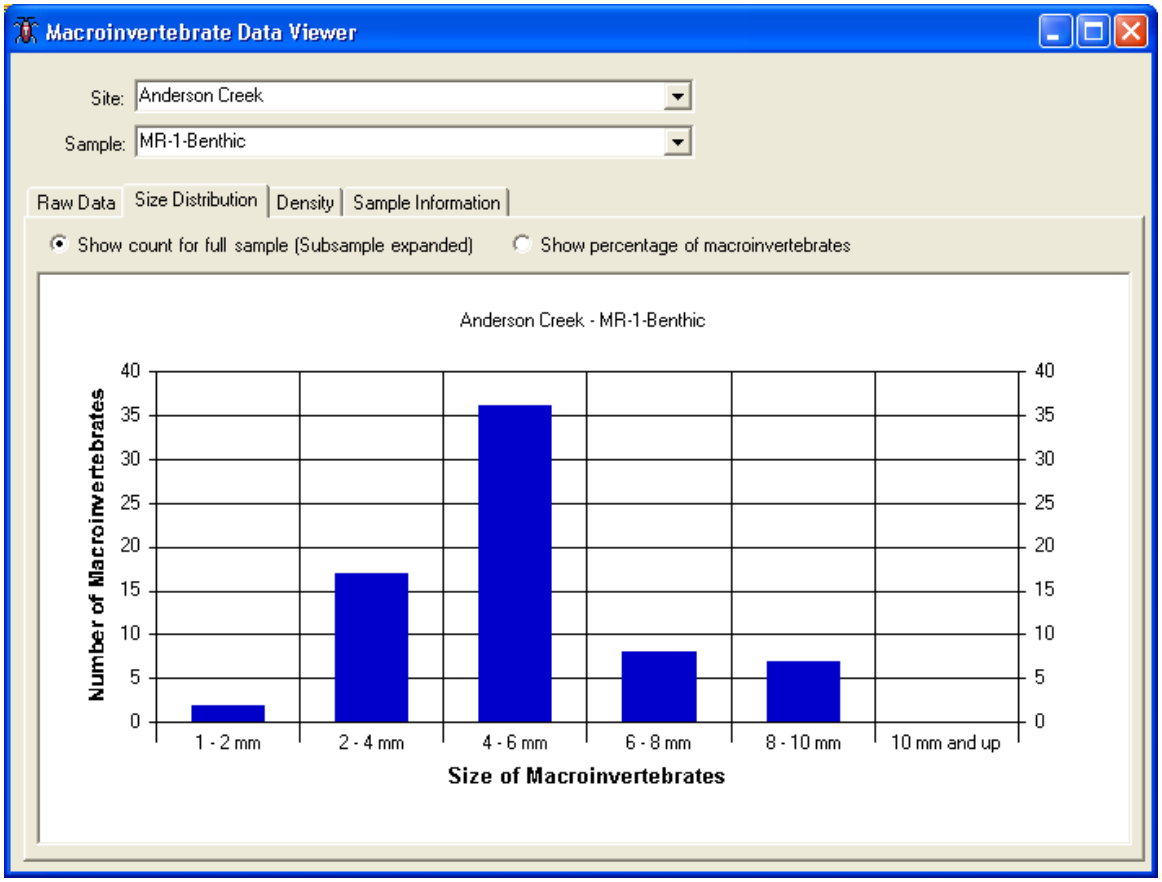
The Raw Data tab displays the data in spreadsheet format by size class and group. If a sample was collected as a "subsample", the user can view the data as it was collected by selecting 

or as the subsample extended to full sample size by selecting

. If "Fraction subsampled" is 100.00%, both selections will display the same spreadsheet.

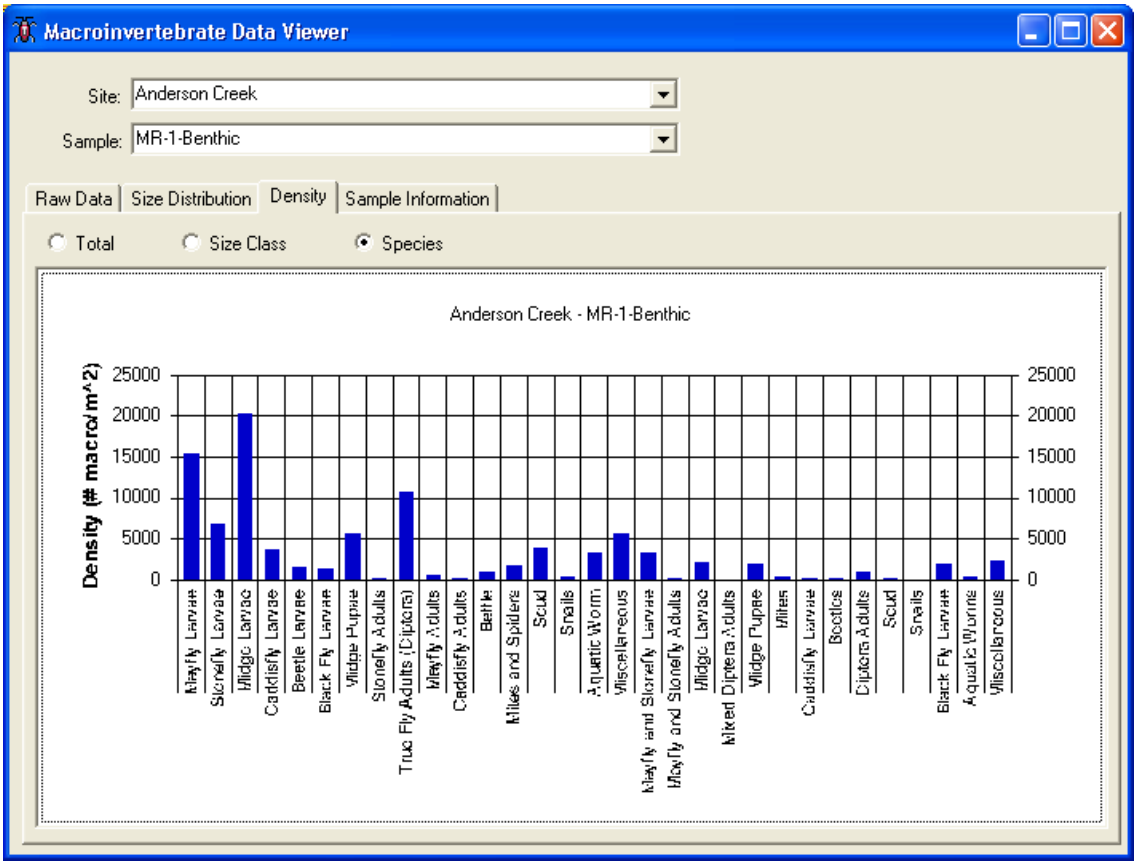
### *Size Distribution Plots*

Click on the Size Distribution tab. The user can choose to view a plot of the number of macroinvertebrates present in each size class (as shown below) by selecting (•) Show count for full sample (Subsample expanded). Alternately, a plot of the percent of macroinvertebrates in each size class can be viewed by selecting ( ) Show percentage of macroinvertebrates.



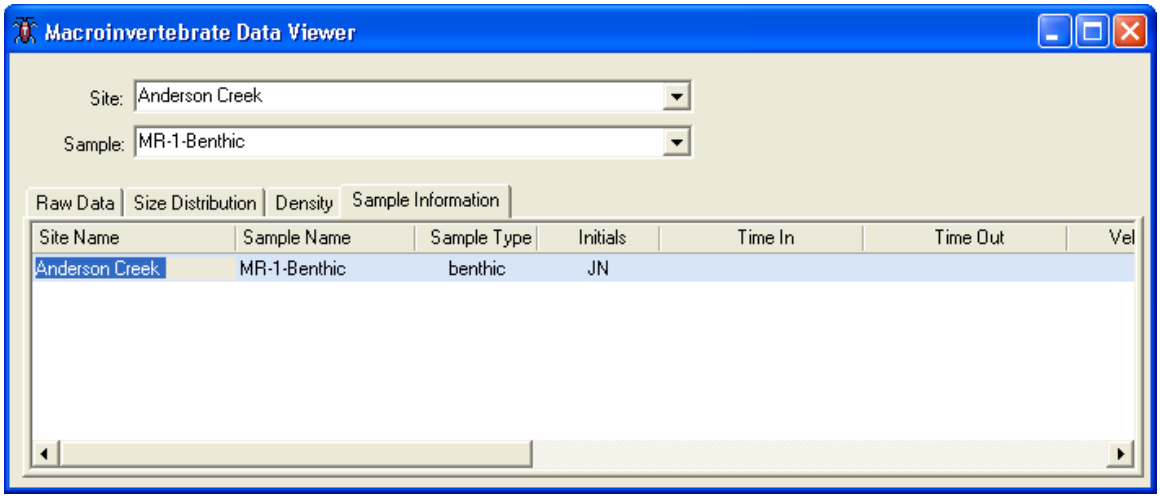
### *Macroinvertebrate Density Plots*

Click on the Density tab to view the density of macroinvertebrates in a sample. Select Total to view a plot of the total density of macroinvertebrates in the sample. Choose Size Class to view a plot of the density of macroinvertebrates in each size class. Select Species to view a plot of macroinvertebrate density by species (as shown below).



### *Viewing Sample Information*

Click on the Sample Information tab to view a summary table of information pertaining to sample collection, including: site name, sample name, sample type, initials of sampler, time in and out, velocity in and out, site coordinates, net area, bed area and mesh size. Use the horizontal scroll bar at the bottom of the window to view all information for a sample.

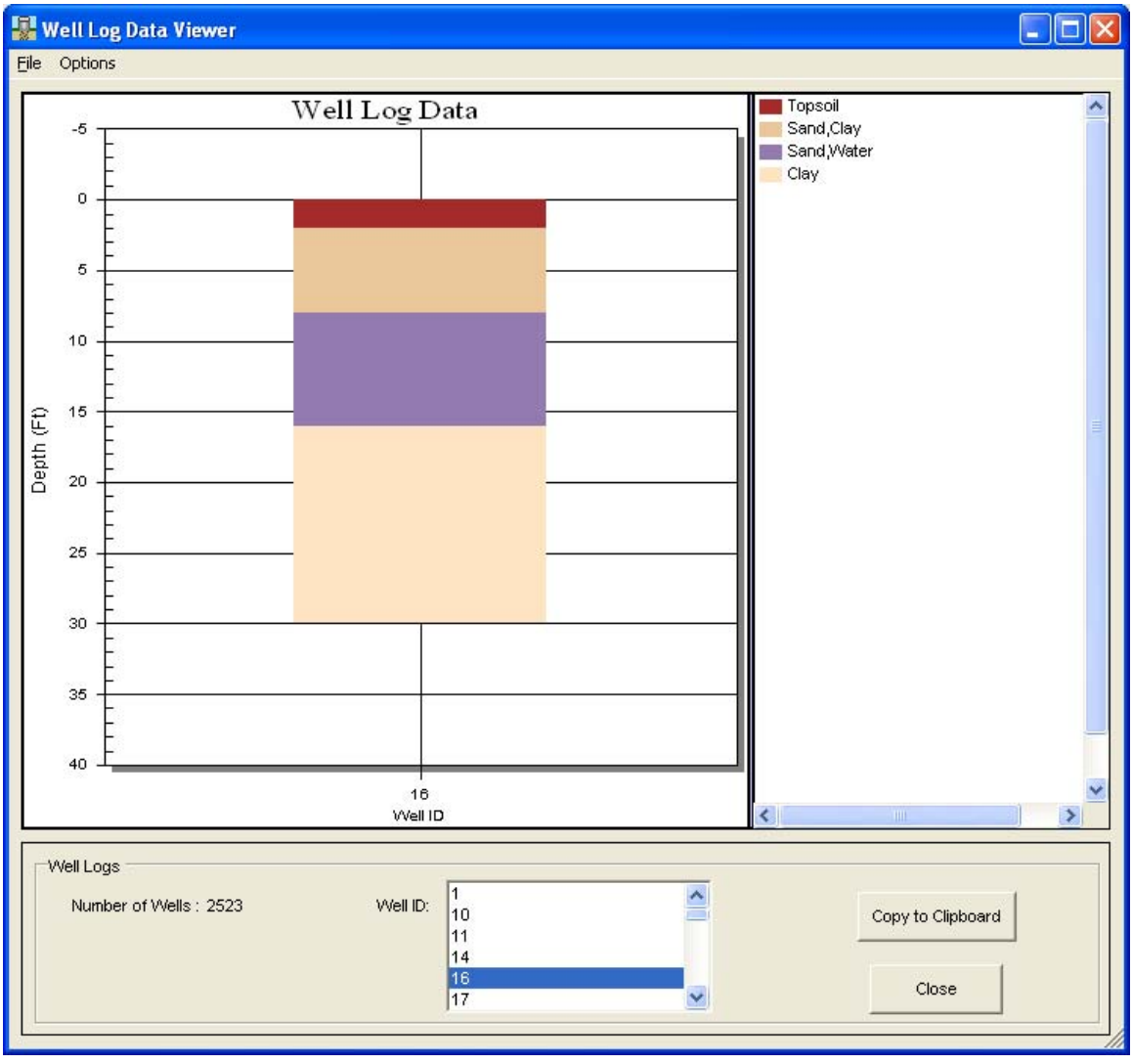


# Well Log Data Viewer Plug-in

The Well Log Data Viewer Plug-in to the DSS allows users to view well graphs and scanned well report documents. A database of well data is linked to a geographic coverage showing the locations at which water well reports and well log data are available.

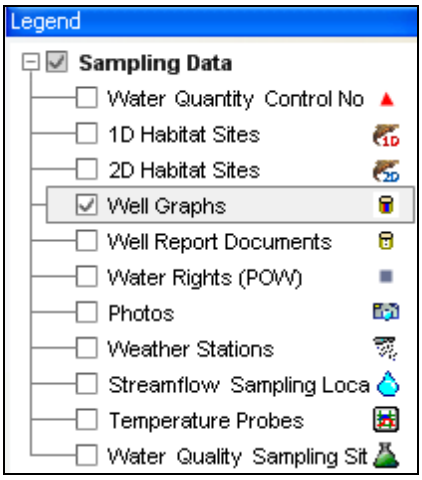
## Viewing well graphs

Well log data is displayed on a bar graph that shows well log depth and the thickness of each soil/rock layer. A color-coded key lists the soil/rock types presented on the graph.



### *Selecting Well Graph Points from the Map*

First make sure that the Well Graphs theme is turned on (checked) and active (selected).




Next, zoom on the map to the area of interest using the pan and zoom tools. Using the select tool , locations can be selected for viewing by either clicking on individual points or by clicking and dragging a box around several locations. Selecting Well Graph points from the map will load the well graph data for those selected locations into the Well Log Data Viewer, shown above.

### *Viewing Well Graphs via the Well Log Data Viewer Toolbar Button*

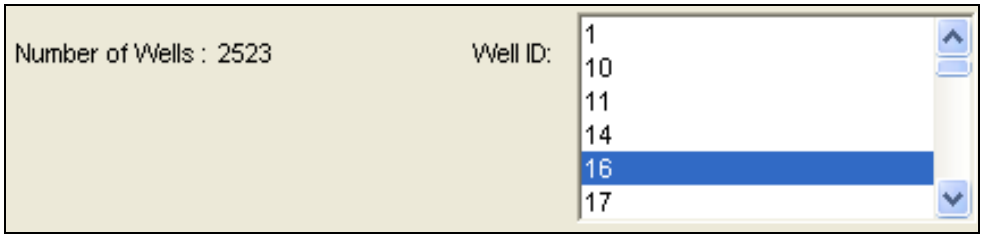
As an alternative to selecting points using the map, clicking the Well Log Data Viewer toolbar button opens the Well Log Data Viewer and loads the entire database of well log data into the Well Log Data Viewer as shown above. Users can browse the entire database of well log graphs without interacting with the map. However, because the Well Log Data Viewer toolbar button is also used to access well report documents, the Well Graphs theme must be active before clicking on the toolbar button.

## Well Graph viewing options

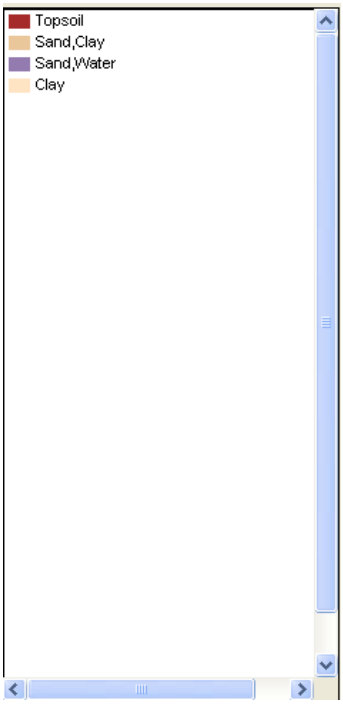
The Well Log Data Viewer can display data for one well or several wells on one graph. Options and features of the Well Log Data Viewer pertaining to well graphs are discussed in the following sections.

### *Viewing a Single Well Graph*

A single well graph can be displayed in the graph window by selecting one Well ID from the window. The scroll bar on the right side of the window allows the user to look through all the available well graphs. The number of wells available for viewing is shown to the left of the Well ID window.



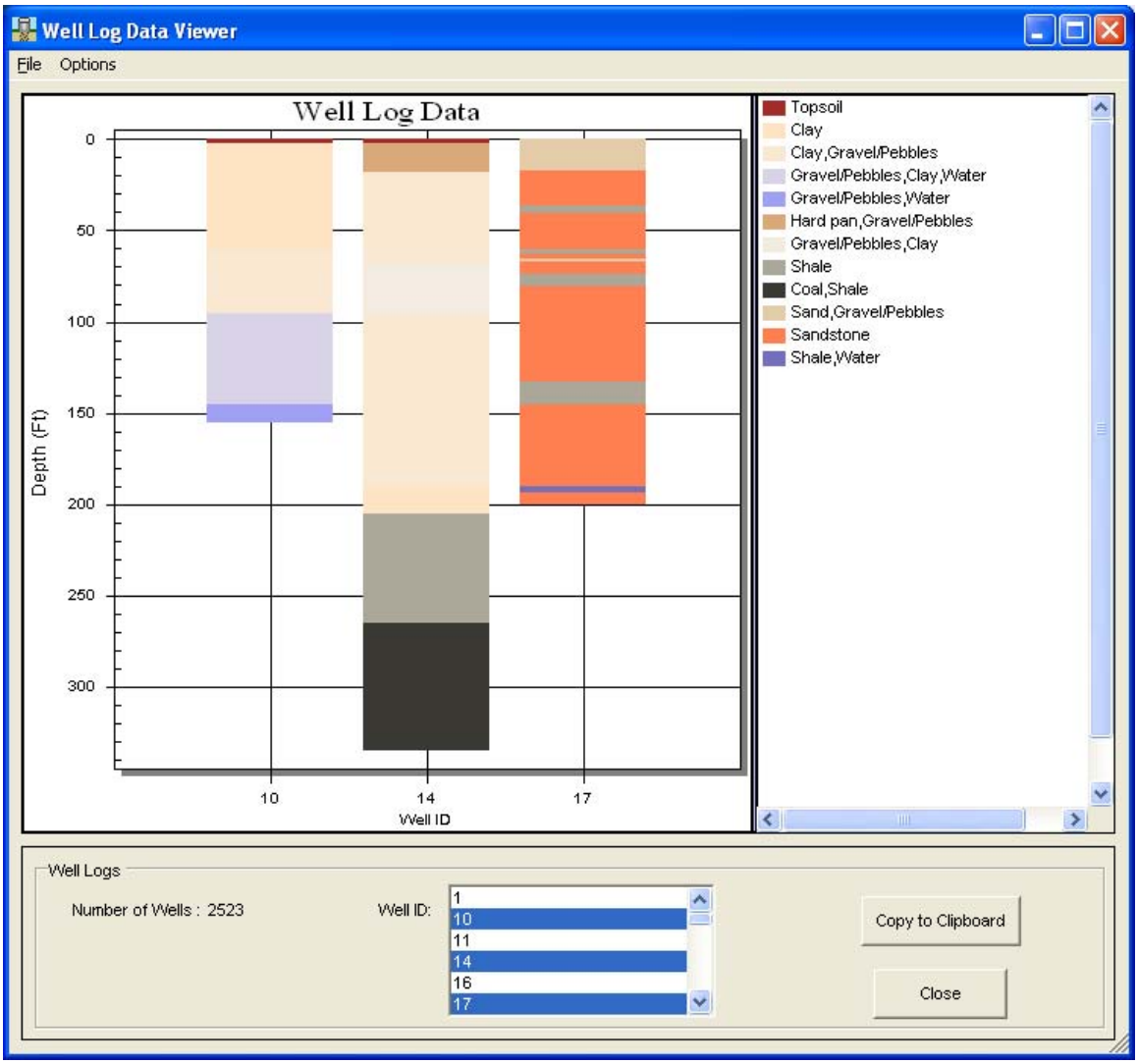

Once a well is selected from the list, the data is plotted on a bar graph and a color-coded key for the graph is displayed to the right of the graph in the adjacent window (as shown above in the Viewing Well Graphs section).




### *Viewing Multiple Well Graphs*

More than one well graph can be displayed on the bar graph at one time by selecting more than one Well ID number from the list in the Well ID window as shown below. To select more than one well, click on the well ID number for the first well to be viewed

then hold the “Ctrl” (control) key on the computer keyboard while clicking on each additional well ID. If the desired wells are listed contiguously, click on the first well ID then hold the “Shift” key on the computer keyboard and select the last well ID desired. Selected well ID numbers will be highlighted in the window. Graphs for selected wells will appear in the graph window and the associated key to soil/rock layers will update automatically.



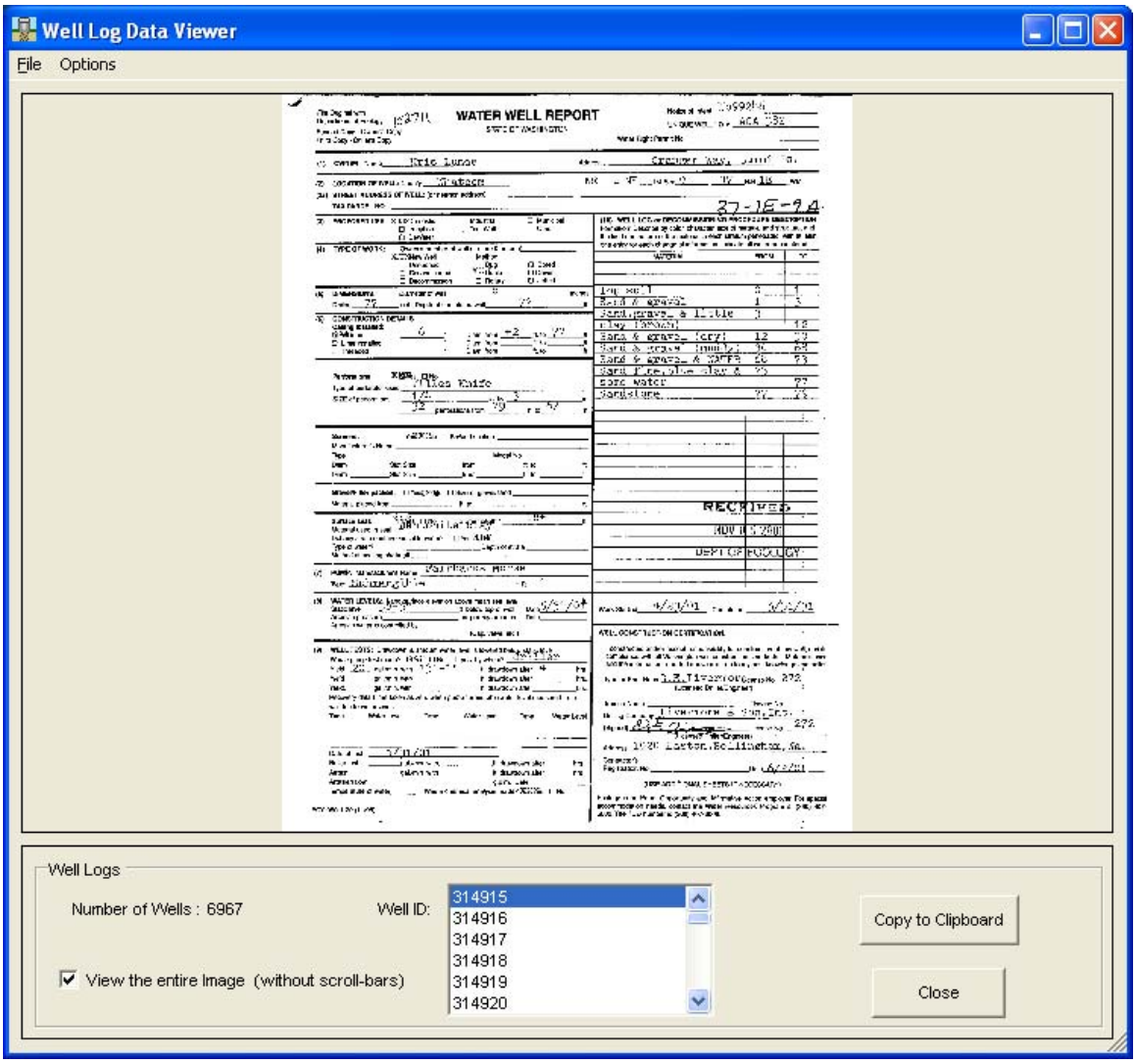

### *Exporting Graphs*

Click on the

button to add the graph and associated key to the “clipboard”. From the clipboard, the graph and key can be pasted into a report or other document and printed.

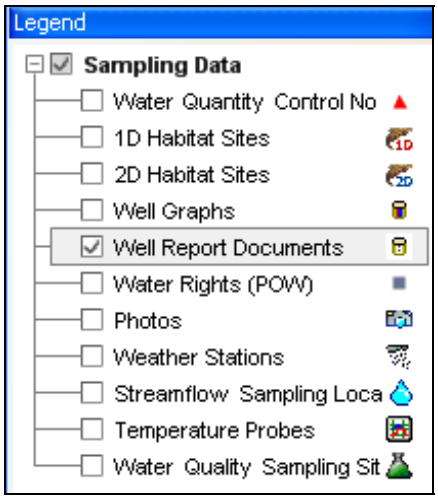
# Viewing Well Report Documents

Scanned well report documents can be viewed using the Well Log Data Viewer.




### *Selecting Well Report Document Points from the Map*

First make sure that the Well Report Documents theme is turned on (checked) and active (selected).




Next, zoom on the map to the area of interest using the pan and zoom tools. Using the select tool , locations can be selected for viewing by either clicking on individual points or by clicking and dragging a box around several locations. Selecting Well Report Document points from the map will load the well report documents for those selected locations into the Well Log Data Viewer, shown above.

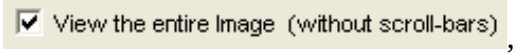
***Viewing Well Report Documents via the Well Log Data Viewer Toolbar Button***

As an alternative to selecting points using the map, clicking the Well Log Data Viewer toolbar button opens the Well Log Data Viewer and loads the entire database of well report documents into the Well Log Data Viewer as shown above. Users can browse the entire database of well report documents without interacting with the map. However, because the Well Log Data Viewer toolbar button is also used to access well graphs, the Well Report Documents theme must be active before clicking on the toolbar button.

## Well Report Document Viewing Options

Well report documents can only be viewed one at a time. The viewer window refreshes automatically as well ID numbers are selected from the list.

***Viewing Entire Well Report Document Image***

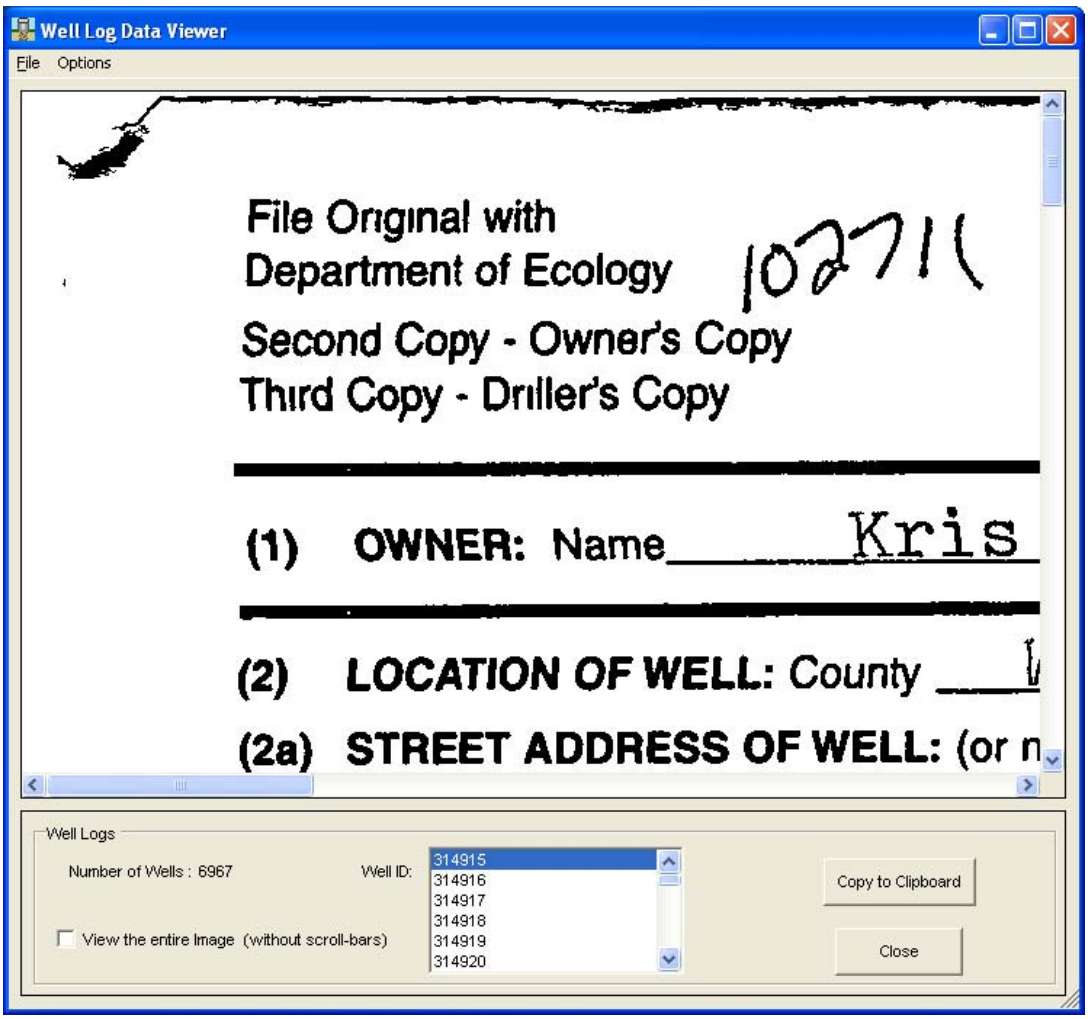
When checked, 

, the entire document is shown in the viewing window as shown above. The number of well report documents available for viewing is shown in the section below the well report document window. Well ID numbers for the sites loaded into the Well Log Data Viewer are listed in the Well ID window.

### *Viewing Enlarged Image of Well Report Document*

When the box

is left unchecked, the image is shown larger than actual size and scroll bars along the side and bottom of the window allow the entire document to be viewed.




The number of well report documents available for viewing is shown in the section below the well report document window. Well ID numbers for the sites loaded into the Well Log Data Viewer are listed in the Well ID window.

### *Exporting Documents*

Click on the

button to add the image of the scanned document to the clipboard. From the clipboard, the image can be pasted into a report or other document and printed.

# Watershed Characterization Report Generator

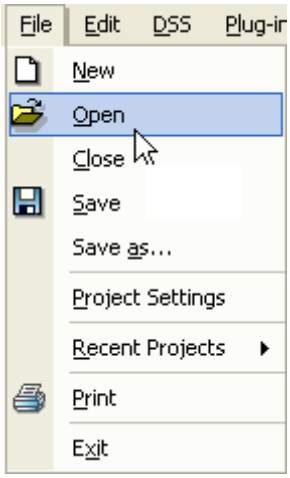
The Watershed Characterization Report Generator is a tool used to summarize a variety of GIS data and database information into a readable report. Output reports can be written to Adobe Acrobat, Microsoft Word, Microsoft Excel, and Rich Text formats.

The datasets that the generator uses to gather and generate data are predefined for each watershed the report generator is used with; data comes from the MapWindow project that is currently loaded. Therefore, in order to use the Watershed Characterization report, a specific project file with the proper data must be loaded into MapWindow.

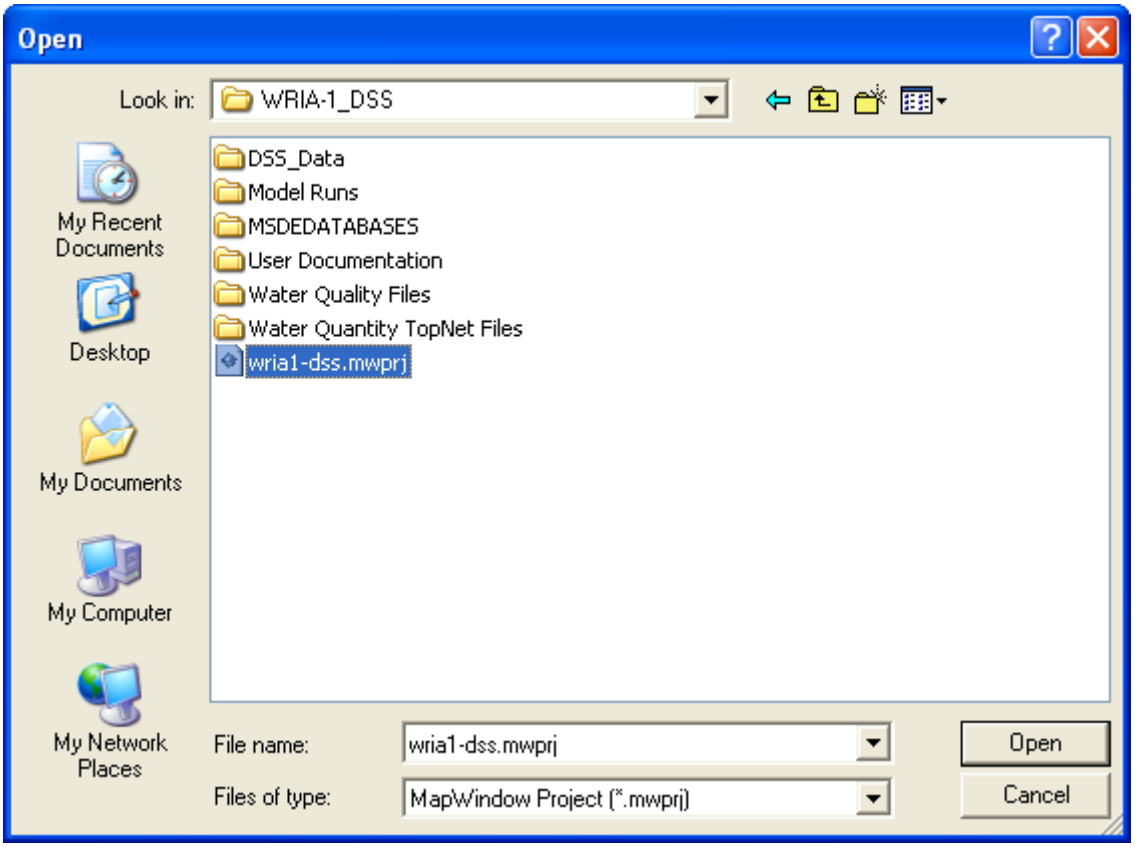
Once the Watershed Characterization generator has loaded, the user may select which drainages should be included in this report. Maps generated will cover only the selected regions, and data will be collected only for these drainages. Once the desired drainages have been selected, the user needs only to click "Launch Report" and wait while the report is created.

### *Opening a MapWindow Project*

Since the Report Generator relies on GIS data defined in a specific project file, first ensure that the proper project file has been opened. Do this by clicking the File menu, then Open as shown below.



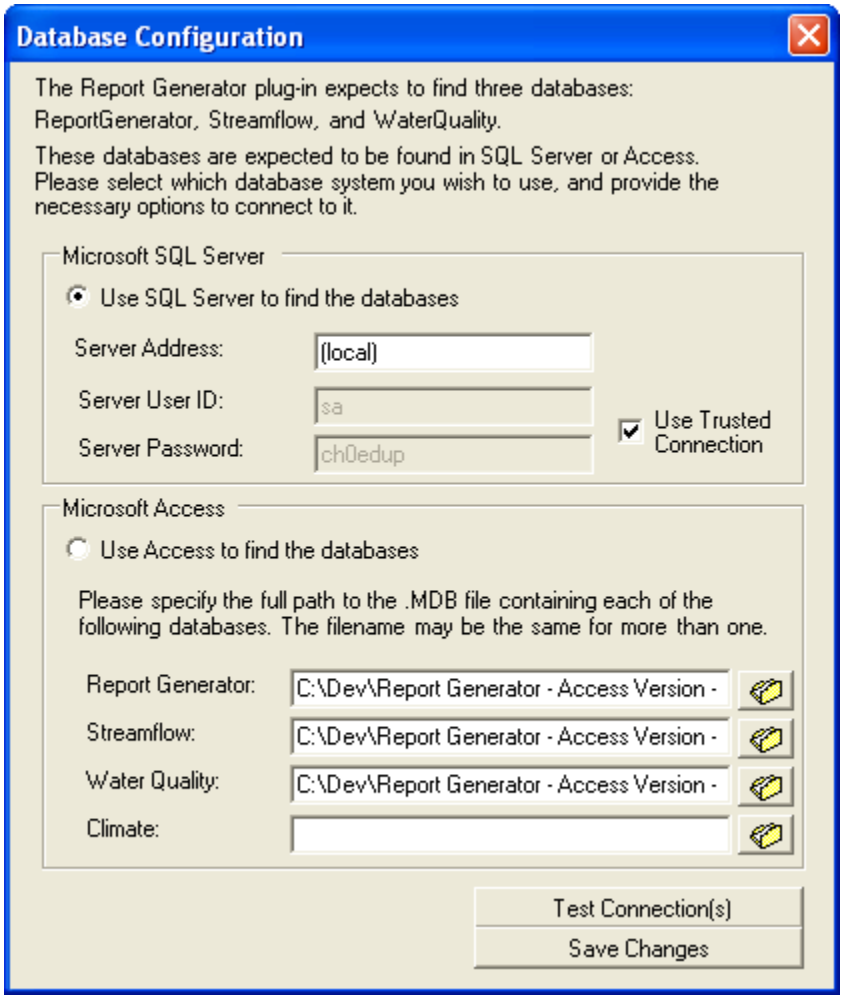

You may also open a project by clicking on the open button located on the toolbar. An Open dialog box will appear, as shown below, allowing you to choose a project.



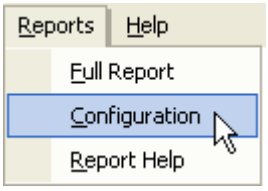

You may get a warning message that one or more databases could not be found. If so, the following section titled Changing the Database Configuration, will help you fix this problem. If you do not get an error message, you may proceed to the section titled Starting the Report Generator Plug-in.

### *Changing the Database Configuration*

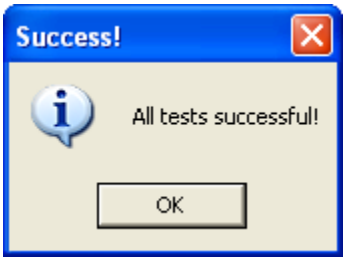
If one or more of the databases for the Report Generator cannot be found, you will be presented with the screen below.


Database Configuration
The Report Generator plug-in expects to find three databases:
ReportGenerator, Streamflow, and WaterQuality.
These databases are expected to be found in SQL Server or Access.
Please select which database system you wish to use, and provide the necessary options to connect to it.
Microsoft SQL Server
Use SQL Server to find the databases
Server Address:
(local)
Server User ID:
sa
Server Password:
ch0edup
Use Trusted Connection
Microsoft Access
Use Access to find the databases
Please specify the full path to the .MDB file containing each of the following databases. The filename may be the same for more than one.
Report Generator:
C:\Dev\Report Generator - Access Version -
Streamflow:
C:\Dev\Report Generator - Access Version -
Water Quality:
C:\Dev\Report Generator - Access Version -
Climate:
Test Connection(s)
Save Changes

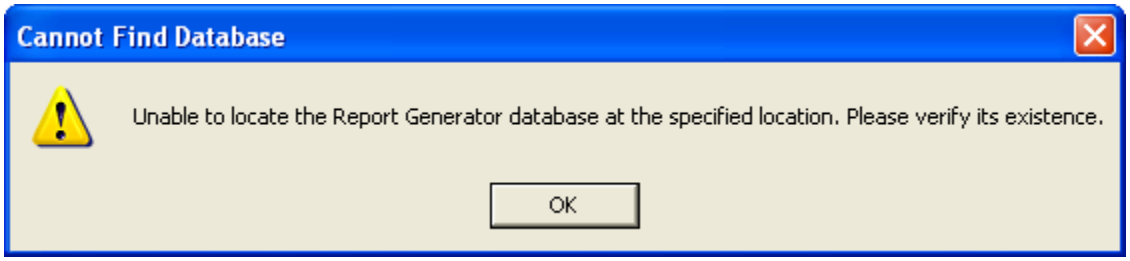
If you wish to change your database configuration at any time, the screen may also be accessed by clicking the Reports Menu, then choosing Configuration, as shown below.




On the Database Configuration screen, select the method (SQL or Access) you would like to use to find the databases. Fill in either the security details such as username and password (contact your administrator if you don't know these), or the paths to the access databases. You may verify that the settings are correct by pressing the **Test Connection(s)** button. If the settings are correct, you will see this screen:



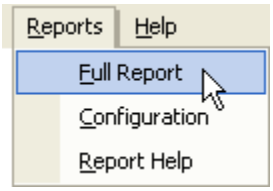

You may then proceed with using the report generator. If the databases could not be found with the specified information, you will see a message similar to this:


Cannot Find Database
Unable to locate the Report Generator database at the specified location. Please verify its existence.
OK

In this case, verify the settings and try again. If you are unable to connect to the database(s), contact your system administrator. Once you have successfully selected the databases you wish to use, press the **Save Changes** button to continue.

### *Starting the Report Generator plug-in*

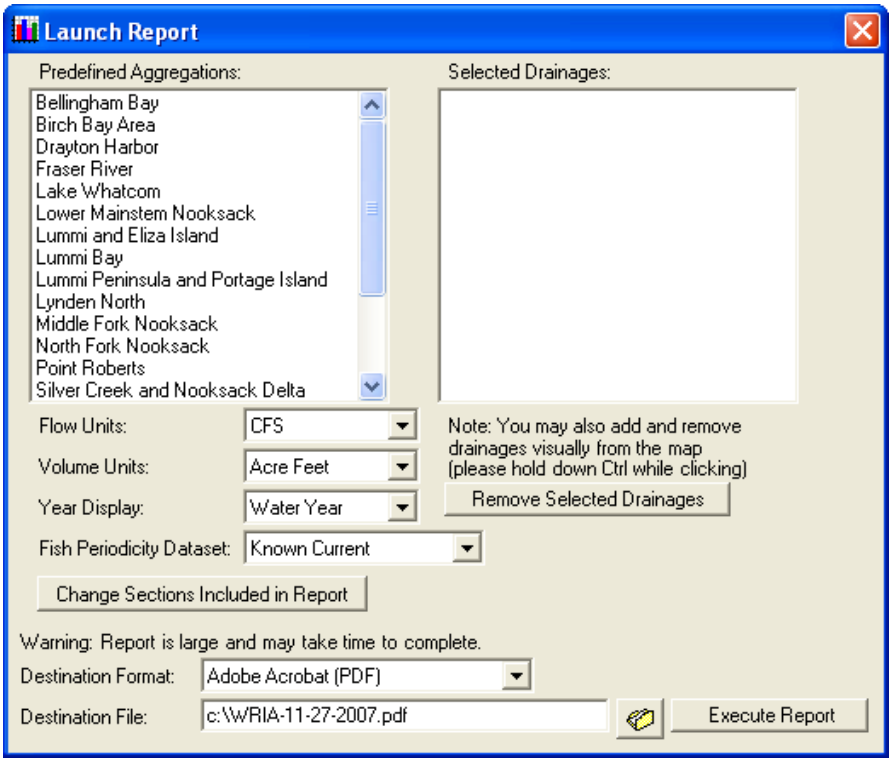
Next, start the Report Generator plug-in by clicking the Reports menu, then selecting Full Report as shown below. (Note: do not use this menu option for the time being; instead use the icon mentioned below.)



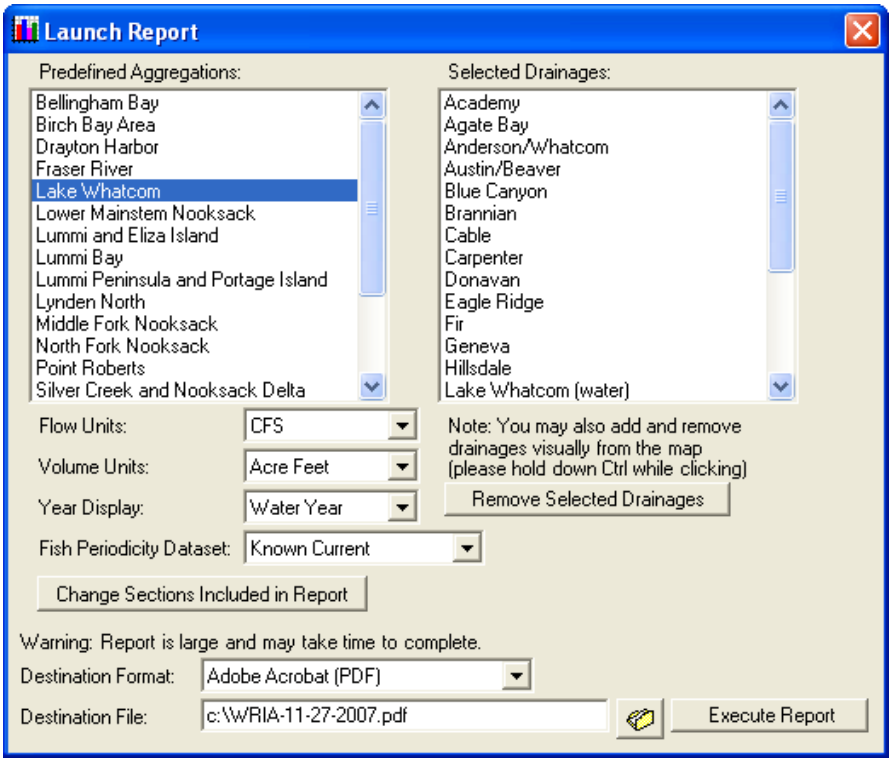

You may also start the plug-in by selecting the graph icon from the toolbar.

### *Generating a Report*

Once the "Launch Report" screen is visible, as shown below, you may select options for the report. This window will look differently, depending on the project you have opened.

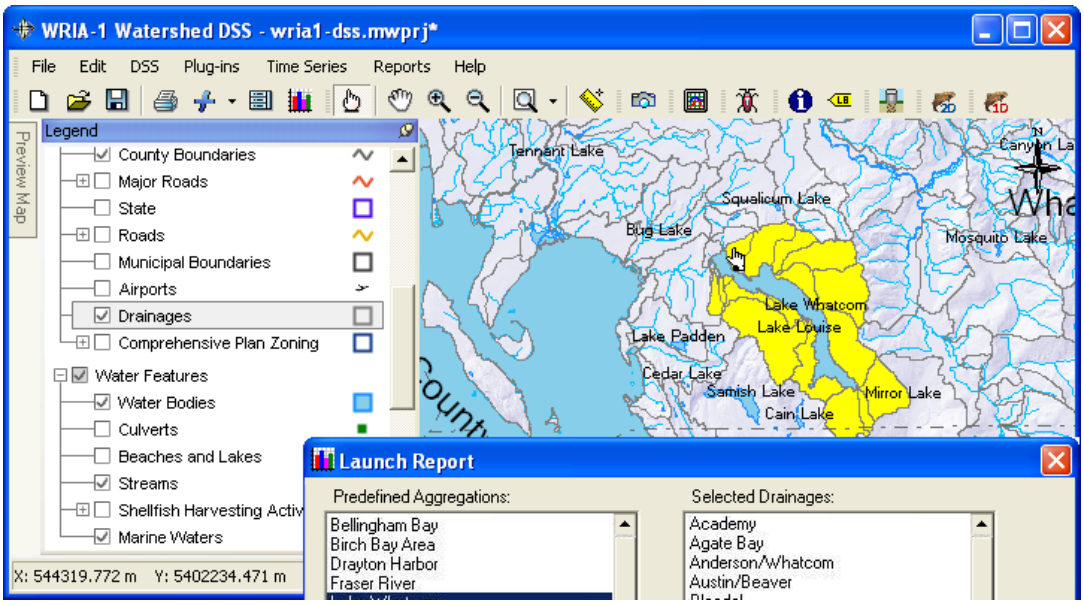



On the left-hand side is a list of predefined aggregations. These are collections of drainages that are commonly grouped together. Click on one of these aggregations, and you'll see the drainages that are associated with this aggregation fill the list on the right.

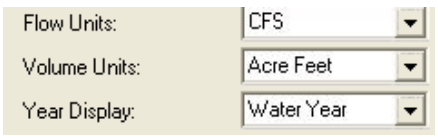


You may also move the Launch Report window out of the way of the main MapWindow screen, and select drainages using the mouse. To do this, the Drainages layer must be selected and you must be using the Select Shapes tool.

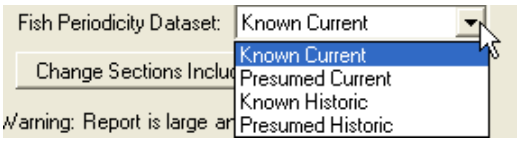
Hold the "Ctrl" key while selecting multiple drainages with the mouse; clicking a drainage that's already selected will deselect it. As drainages are selected or deselected from the map, the list in the Launch Report window will be updated. Note that doing this does not permanently change the contents of the aggregation, but only selects drainages for the report. You may also select drainages to report on without first selecting an aggregation, by using the map to select drainages.



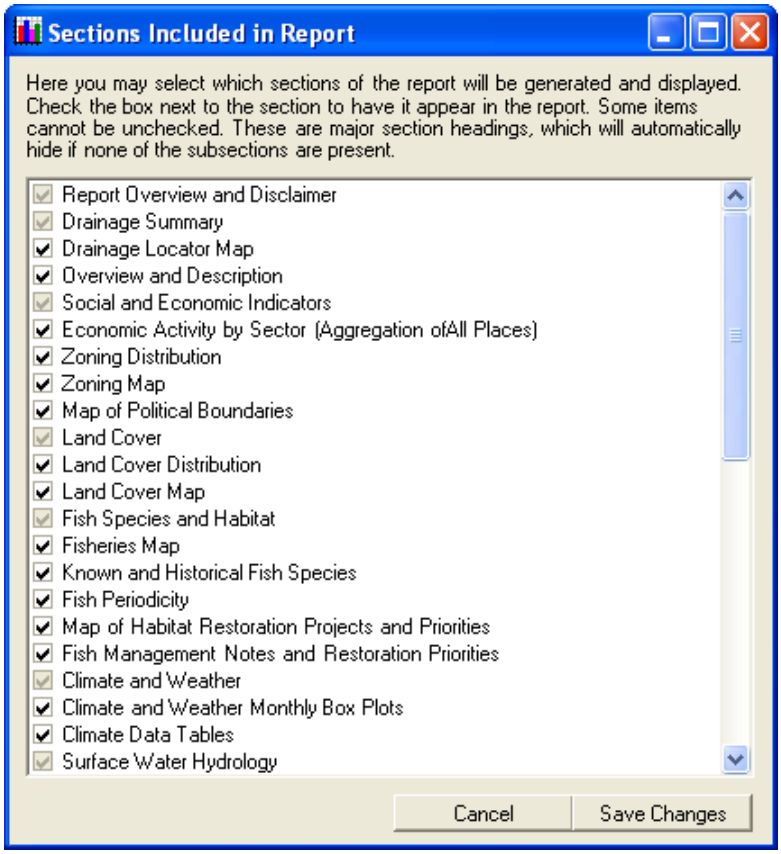

Once the drainages that you wish to include in the report appear in the Selected Drainages list, ensure that Flow Units, Volume Units and Year Display are all set according to your preference. This may be done by clicking on the down arrow next to the units and choosing the preferred option from the list.




Fish Periodicity Dataset refers to which of the four available fish datasets should be included in the Fish Periodicity section of the report.

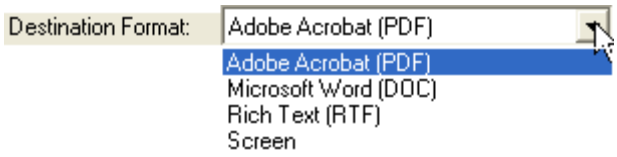


To choose which sections are included in the report, you may click the "Change Sections Included in Report" button. This will display the screen shown below.


Sections Included in Report
Here you may select which sections of the report will be generated and displayed. Check the box next to the section to have it appear in the report. Some items cannot be unchecked. These are major section headings, which will automatically hide if none of the subsections are present.
Report Overview and Disclaimer
Drainage Summary
Drainage Locator Map
Overview and Description
Social and Economic Indicators
Economic Activity by Sector (Aggregation ofAll Places)
Zoning Distribution
Zoning Map
Map of Political Boundaries
Land Cover
Land Cover Distribution
Land Cover Map
Fish Species and Habitat
Fisheries Map
Known and Historical Fish Species
Fish Periodicity
Map of Habitat Restoration Projects and Priorities
Fish Management Notes and Restoration Priorities
Climate and Weather
Climate and Weather Monthly Box Plots
Climate Data Tables
Surface Water Hydrology
Cancel
Save Changes

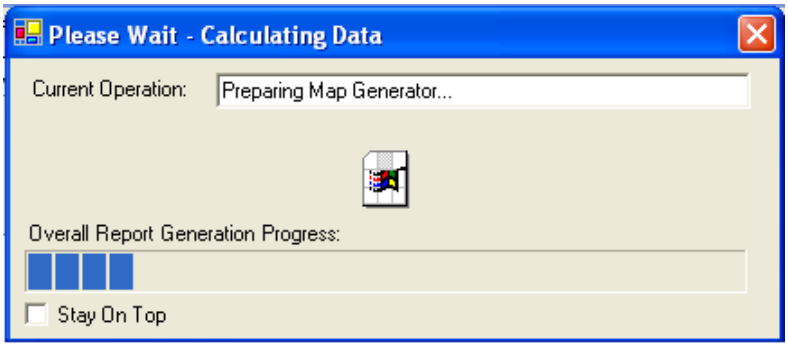
Using this window, you may select which sections are going to be generated for and appear in the report. You cannot uncheck some of the boxes; some of these are required, such as the Report Overview and Disclaimer. Most of the time, when you cannot uncheck a box, it's because the box represents a major section heading. These appear and disappear automatically depending on whether any minor sections appear in that major section. Ensure that you've checked the sections you want in the report, and unchecked those you don't want; then click Save Changes. You will then return to the Launch Report screen.

Finally, select the format that you wish the report to be saved as by clicking the down arrow to the right of Destination Format: and selecting the desire output format as shown below.

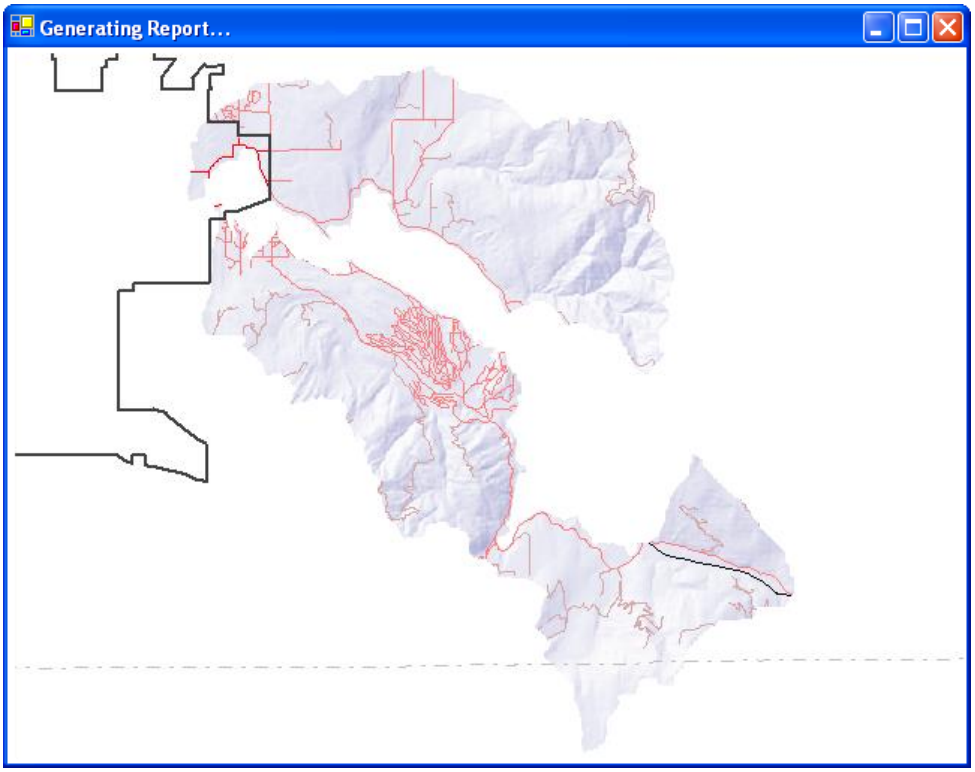


Enter a filename and destination for the report using the box to the right of Destination File. You may click the browse button to navigate to the desired output destination for the report.

Once you have selected all the options you desire, click the **Execute Report** button. While the report is generating, you'll see a progress bar similar to the one shown below.




You may also see maps being created, similar to those shown in the window below.




When the report has finished being generated, it will be displayed. MapWindow may then be closed.

# Appendix A:

# Importing Data into the Master Server

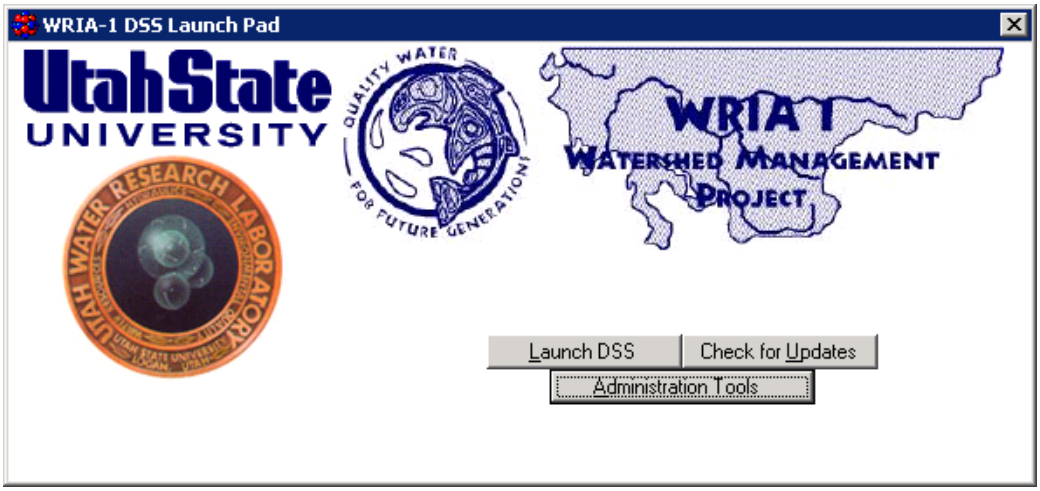
Data is edited directly on the master server using the methods outlined below. This data is propagated to user computers when the user clicks the 'Check for Updates' button on the DBMS, then selects the link indicating that database updates are available.

The data changes are then downloaded to the user's machine. Any records that the user deleted on the local database (i.e., via models) will be recreated; and data that was created on the local machine (again, perhaps by models) will be left intact.

## Editing Data Manually

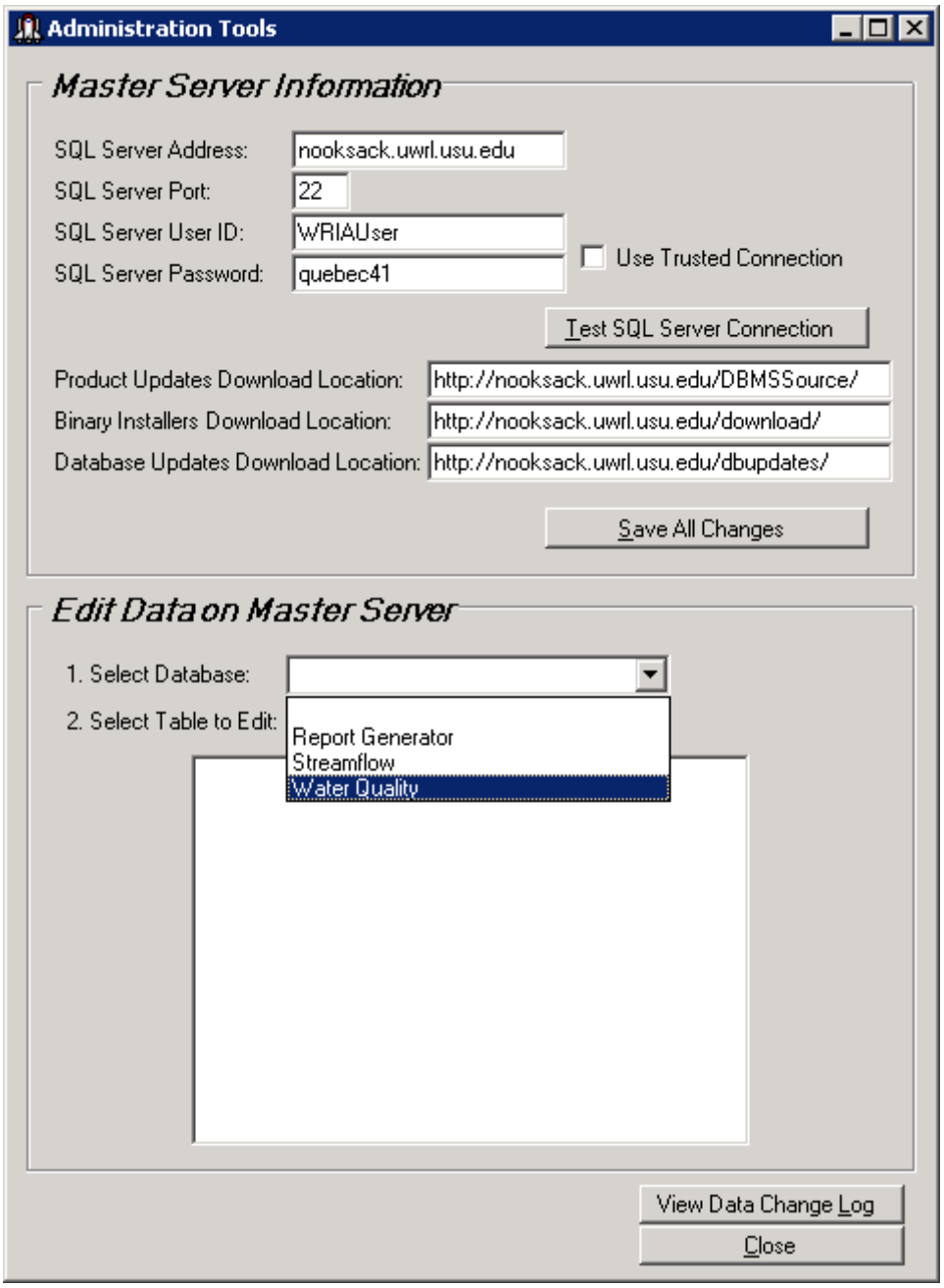
A) Launch the DBMS application. You must have the Administrative Edition to continue, and you must also know the administration password. Terry Holland may be contacted for these items.

B) Click the Administration Tools button:

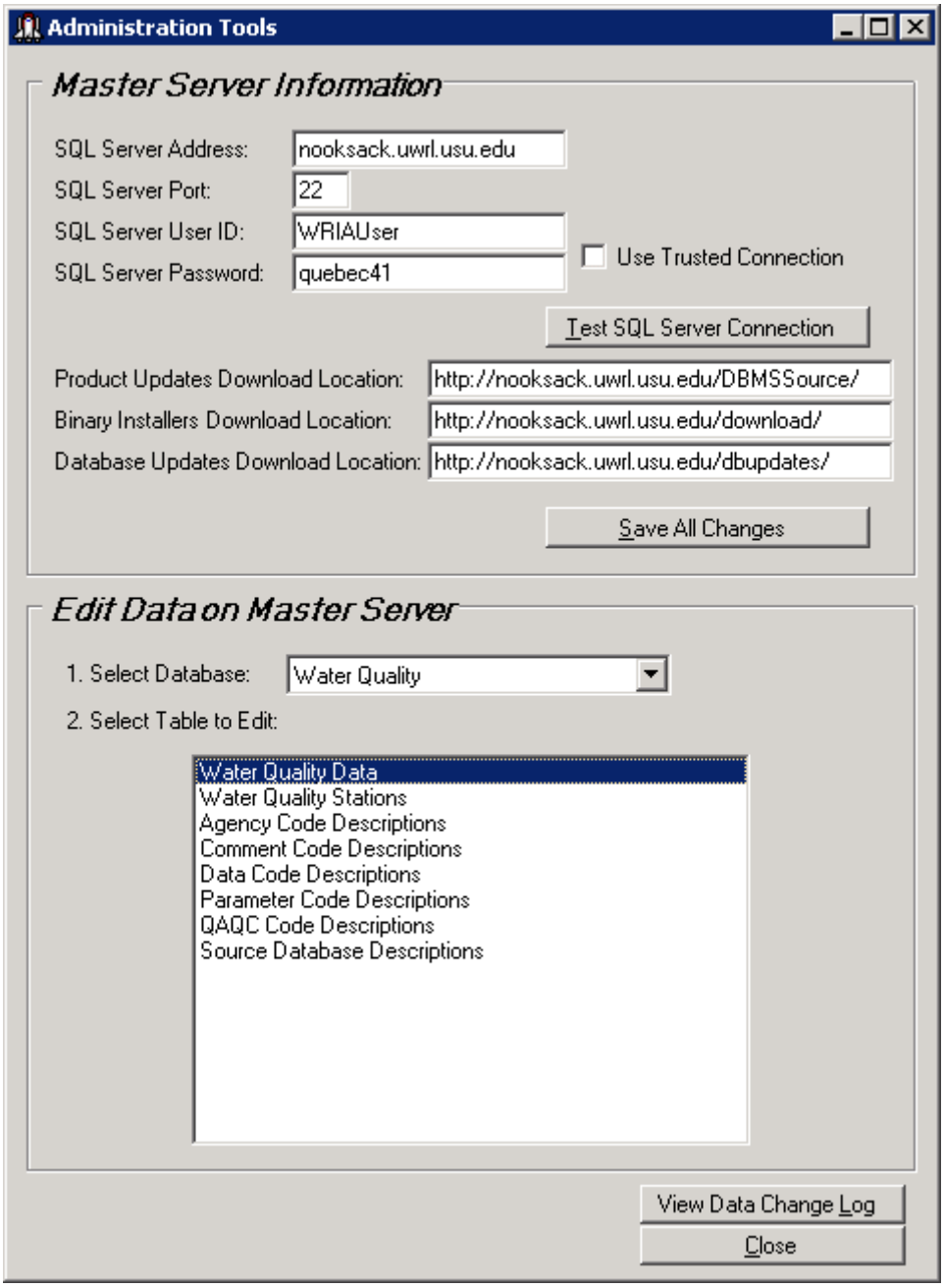



C) Enter the password:

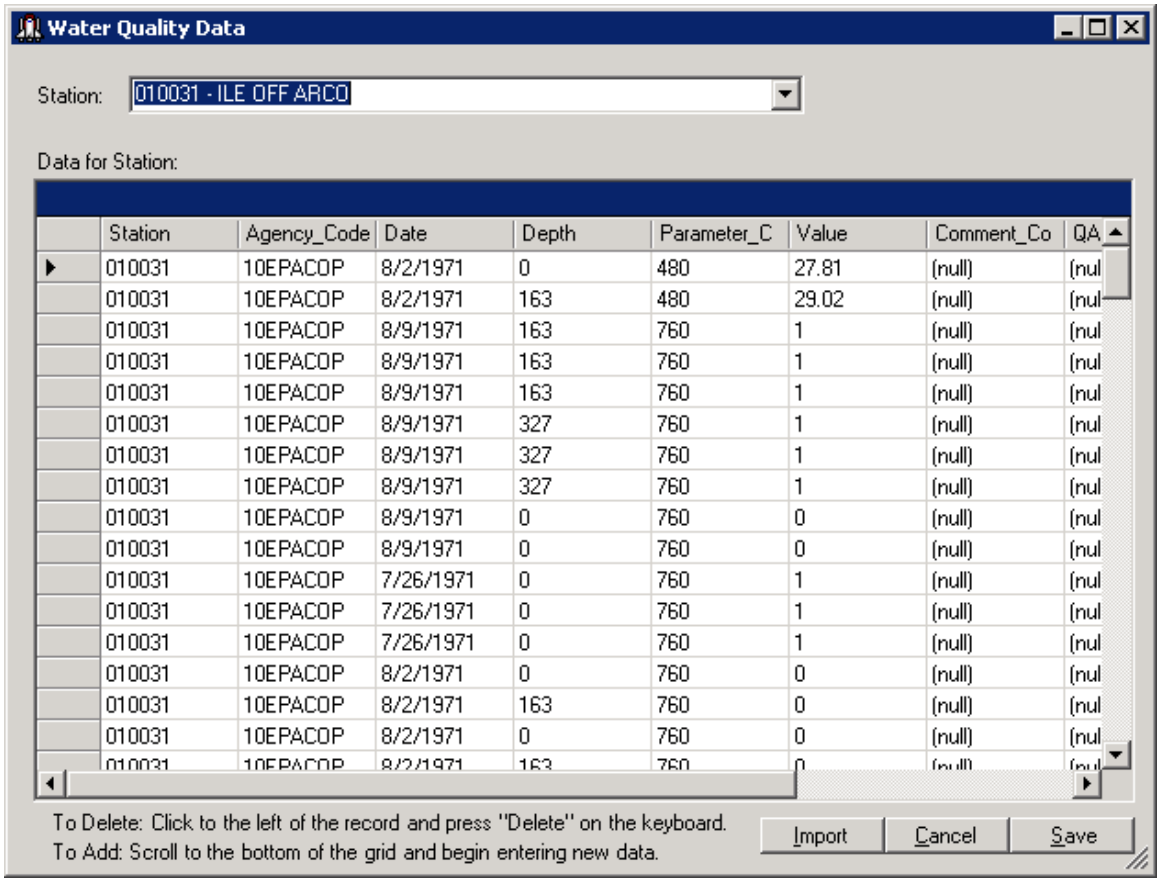
D) Next, you will be presented with the window below. Select the database that you wish to edit, by selecting it from the drop-down list labeled “1. Select Database:”.




E) Next, click the name of the table with contents you wish to edit.




F) Clicking the name of the table to edit as shown above will cause the data editor for that table to be displayed, shown below. For Water Quality and Flow Data tables, you may select a particular station from the drop-down list to view the data for. (There is too much data to show all of it.) If you wish to add or edit stations, edit the appropriate Stations table.


Water Quality Data
Station:
010031 - ILE OFF ARCO
Data for Station:
Station | Agency_Code | Date | Depth | Parameter_C | Value | Comment_Co | QA
010031 | 10EPACOP | 8/2/1971 | 0 | 480 | 27.81 | (null)
010031 | 10EPACOP | 8/2/1971 | 163 | 480 | 29.02 | (null)
010031 | 10EPACOP | 8/9/1971 | 163 | 760 | 1 | (null)
010031 | 10EPACOP | 8/9/1971 | 163 | 760 | 1 | (null)
010031 | 10EPACOP | 8/9/1971 | 163 | 760 | 1 | (null)
010031 | 10EPACOP | 8/9/1971 | 327 | 760 | 1 | (null)
010031 | 10EPACOP | 8/9/1971 | 327 | 760 | 1 | (null)
010031 | 10EPACOP | 8/9/1971 | 327 | 760 | 1 | (null)
010031 | 10EPACOP | 8/9/1971 | 0 | 760 | 0 | (null)
010031 | 10EPACOP | 8/9/1971 | 0 | 760 | 0 | (null)
010031 | 10EPACOP | 7/26/1971 | 0 | 760 | 1 | (null)
010031 | 10EPACOP | 7/26/1971 | 0 | 760 | 1 | (null)
010031 | 10EPACOP | 7/26/1971 | 0 | 760 | 1 | (null)
010031 | 10EPACOP | 8/2/1971 | 0 | 760 | 0 | (null)
010031 | 10EPACOP | 8/2/1971 | 163 | 760 | 0 | (null)
010031 | 10EPACOP | 8/2/1971 | 0 | 760 | 0 | (null)
To Delete: Click to the left of the record and press "Delete" on the keyboard.
To Add: Scroll to the bottom of the grid and begin entering new data.
Import
Cancel
Save

G) As the instructions at the bottom of the screenshot state, you may delete a record by clicking in the grey area of the record to highlight it; then, press delete on the keyboard. To add a record, scroll to the bottom of the grid where the empty line is displayed, and begin entering data in that row. Editing data may be done directly on the grid by clicking in the field to be edited and typing. Additionally, you may sort the data view by a given column by clicking the column name. You may resize columns by clicking between the column names and dragging; the window itself may also be resized.

H) Not all data editors have a station drop-down, as mentioned above, as it is not appropriate for all data editors.

I) Data is saved by clicking the Save button in the right-hand lower corner. If you do not click Save (i.e., just close the window), the changes will not be saved.

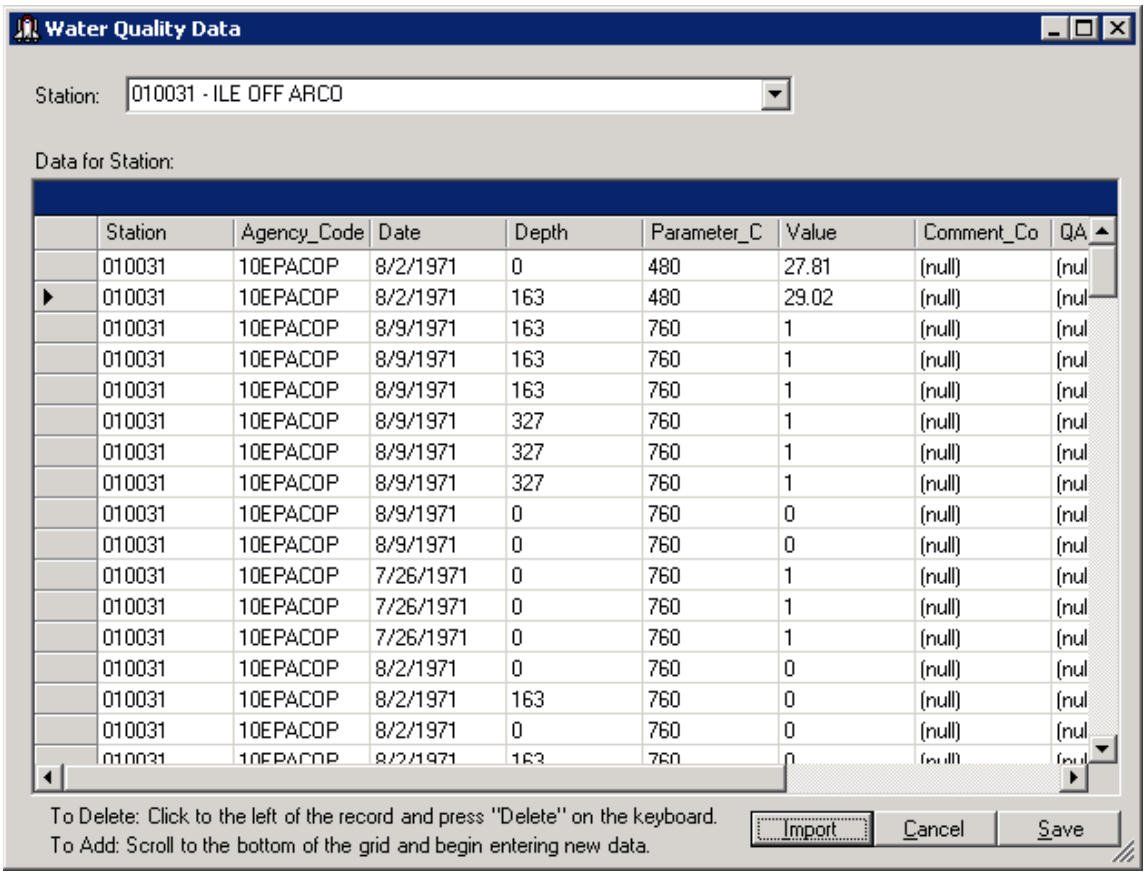
## Importing Data Automatically

A) Data may be imported into the database tables automatically from a delimited text file. This delimeter may be a comma, semicolon, tab character, or pipe (|). The

order in which the fields must appear in the file is displayed on the Import screen, described below. You are responsible for ensuring that your import operation will not create duplicate or invalid data, the importer does not check for duplicate records.

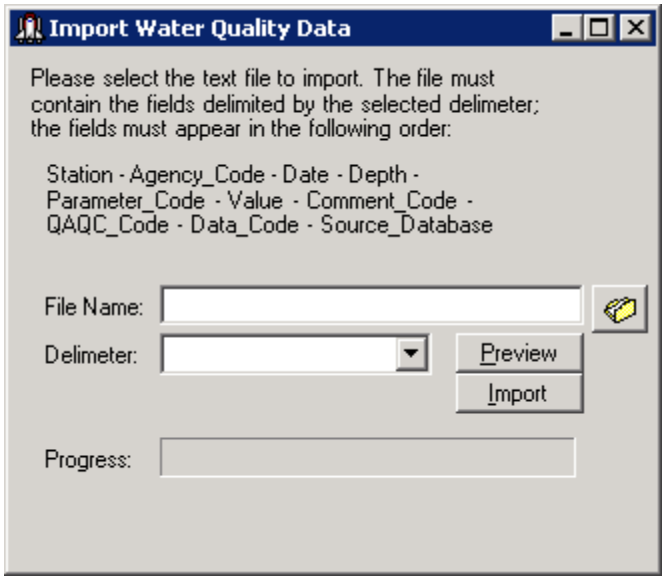
B) First, follow all of the steps in Editing Data Manually, up to step F causing the data editor grid to be displayed.

C) Click the Import button in the lower right-hand corner of the window.

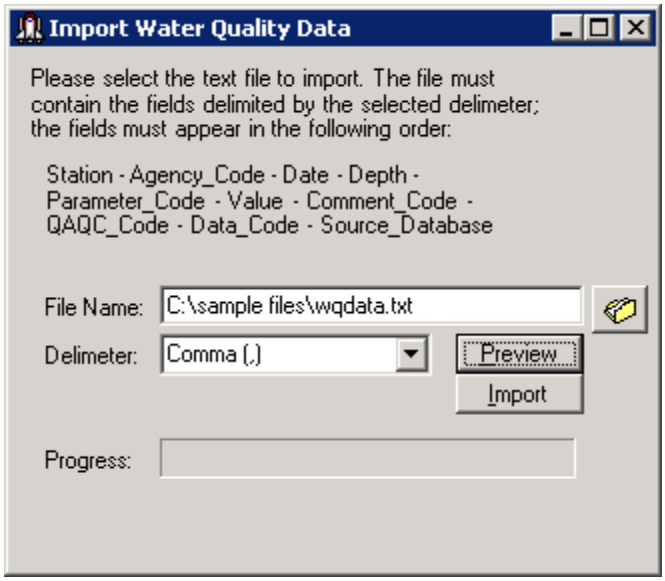

Water Quality Data
Station:
010031 - ILE OFF ARCO
Data for Station:
Station
Agency_Code
Date
Depth
Parameter_C
Value
Comment_Co
QA
010031 10EPACOP 8/2/1971 0 480 27.81 (null)
010031 10EPACOP 8/2/1971 163 480 29.02 (null)
010031 10EPACOP 8/9/1971 163 760 1 (null)
010031 10EPACOP 8/9/1971 163 760 1 (null)
010031 10EPACOP 8/9/1971 163 760 1 (null)
010031 10EPACOP 8/9/1971 327 760 1 (null)
010031 10EPACOP 8/9/1971 327 760 1 (null)
010031 10EPACOP 8/9/1971 327 760 1 (null)
010031 10EPACOP 8/9/1971 0 760 0 (null)
010031 10EPACOP 8/9/1971 0 760 0 (null)
010031 10EPACOP 7/26/1971 0 760 1 (null)
010031 10EPACOP 7/26/1971 0 760 1 (null)
010031 10EPACOP 7/26/1971 0 760 1 (null)
010031 10EPACOP 8/2/1971 0 760 0 (null)
010031 10EPACOP 8/2/1971 163 760 0 (null)
010031 10EPACOP 8/2/1971 0 760 0 (null)
To Delete: Click to the left of the record and press "Delete" on the keyboard.
To Add: Scroll to the bottom of the grid and begin entering new data.
Import
Cancel
Save

D) This will cause the window below to be displayed, which describes the text file format needed in order to import data. You may click the yellow folder icon to browse for the text file. Next, select the field delimiter that your text file uses.

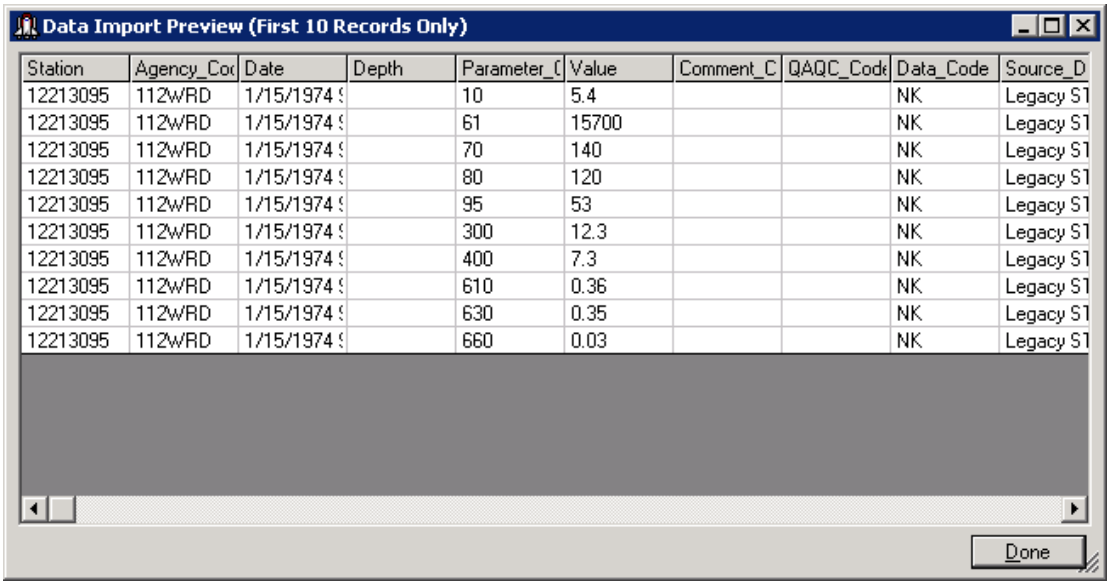
Records are delimited by a new line [carriage return/linefeed]).


Import Water Quality Data
Please select the text file to import. The file must contain the fields delimited by the selected delimeter; the fields must appear in the following order:
Station - Agency_Code - Date - Depth - Parameter_Code - Value - Comment_Code - QAQC_Code - Data_Code - Source_Database
File Name:
Delimeter:
Preview
Import
Progress:

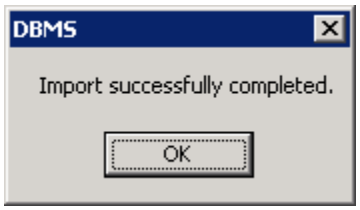
E) Once the text file and delimiter are selected, you may click Preview. *If you selected the wrong delimiter, all data will show up in the first column.*


Import Water Quality Data
Please select the text file to import. The file must contain the fields delimited by the selected delimeter; the fields must appear in the following order:
Station - Agency_Code - Date - Depth - Parameter_Code - Value - Comment_Code - QAQC_Code - Data_Code - Source_Database
File Name:
C:\sample files\wqdata.txt
Delimeter:
Comma (,)
Preview
Import
Progress:

F) Clicking preview will cause the window below to appear. This is intended to show what data is about to be imported. **Only the first 10 rows will be shown.** Once you have made sure that the data about to be imported is correct and is what was intended, close the window returning to the Import window. In this preview window, you may resize the columns by positioning the mouse between the column names and dragging. You may also resize the window itself.




G) After returning to the import window, click the Import button. The progress meter will show the progress through the text file. When the process completes, the Import window will show the number of records imported below the progress bar, and the window shown below will appear:




H) If an error is found in the file, an error message will appear. The system will do its best to help describe the error, or at least what line in the file the error occurs at. Common things to check are that all required values (keys, etc) are supplied, ensure that all fields, even blank fields, are present (ie, for value3 to be blank: value1,value2,,value4). Also check that data types are correct, for example don't try to use "Hello there" as a sample value if sample value is a numeric field.

I) If an error does occur, NO data will be imported; the entire process will be aborted. Data is only written to the database upon successful completion of the entire file.

J) An example of an error during import appears below. In this case, no data was imported to the database because an error occurred. To cause this error, I placed an 'x' where there was a '30', trying to place data of type text into a field of type float.

# Appendix B:

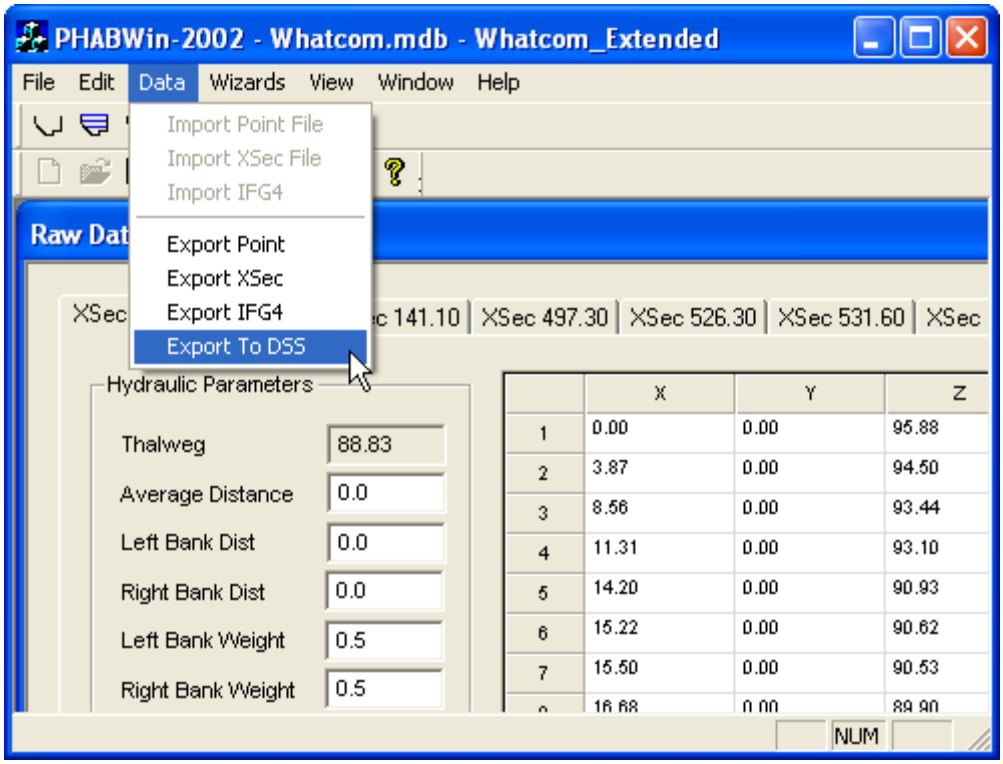
# Importing 1D and 2D data from PHABWin_2002

## Appendix B-1: Importing 1D data from PHABWin_2002

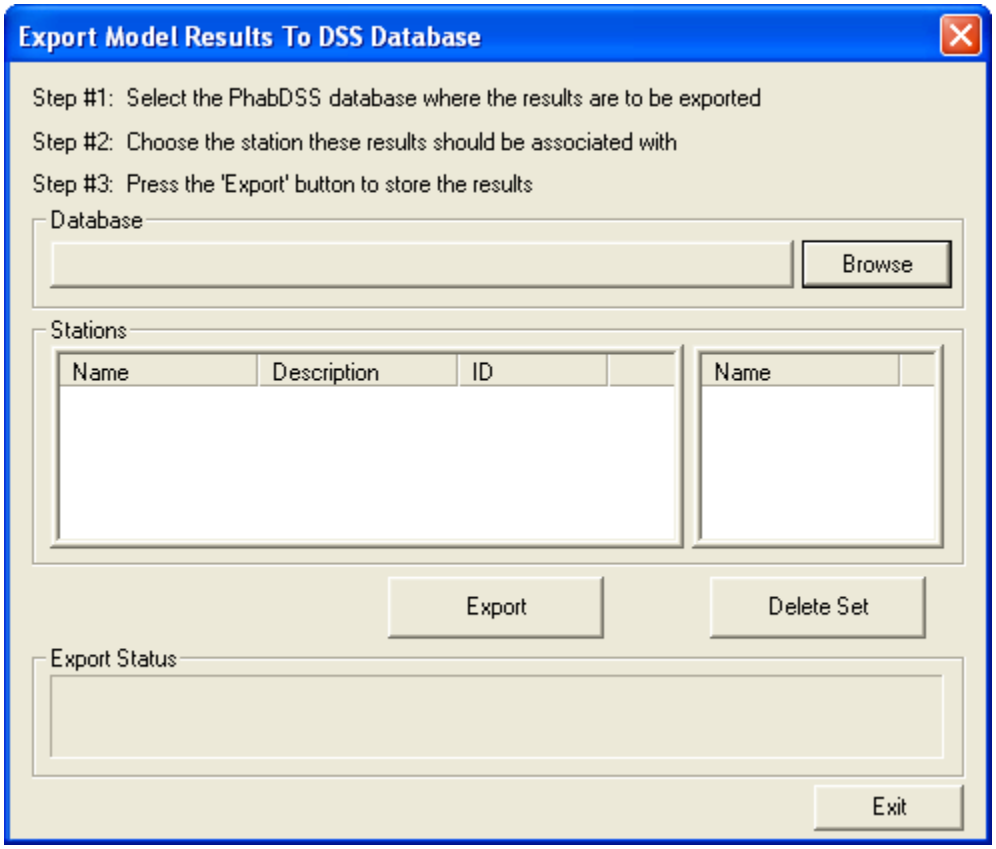
To obtain data to import into the WIRA-1_DSS 1D importer, you must first run the USU PHABWin_2002 program. This includes running the following programs within PHABWin_2002 to enable the 'Export to DSS' option:

1. Both Water Surface Level (WSL) and the Velocity model
2. The Habtae and Habef (Habitat Models)

Once these modules have been run in PHABWin_2002, the Export to DSS option becomes available under the Data drop down menu button as seen below.



Click on the Export to DSS option on the drop-down menu and the following screen will appear.




First, select the database to which the model results will be saved by clicking on the Browse button and navigating to the appropriate DSS database location. Next, choose the station with which these results should be associated by highlighting the appropriate path given in the window. Once the required path has been highlighted under the 'stations' window, click on the Export button. A new window will appear asking the user to 'Enter a new name for this result set'. Give the data set a name and press OK. The database has been exported successfully.

---

# Appendix B-2: Importing 2D Data into the 2D Habitat Viewer

Below are step-by-step instructions for importing data into the 2D Habitat Viewer. Refer to the next section titled "Using the 2D Data Importer" for the sections using the importer tool itself; that section provides step-by-step instructions for using the importer tool,

including screen shots. No screen shots are given for using Access to edit the database directly.

## Folder Structure

The folder structure that exists for the 2D Viewer is given below. To add new data to the 2D Viewer, create a new site folder in the 2D Data folder. The existing 2D Data folder contains the following folders for existing site data:

- HSC
- *Fish Distribution*
  Inside the Fish Distribution folder create the following folders:
  - *Background layers*
  - *Distribution Layers*
- *A folder for each Station*

(named ##-Site where ## is a two-digit number code)

Inside each station folder, create the following folders:

- *Background Image*
- *FishObs Overlays*
- *Mesh*
  - Flow
  - Habitat
  - Species
- *Miscellaneous*
  Inside each Miscellaneous folder (for each station) create the following folders:
  - Aerial View
  - AVI
  - PDF
  - Photos
- *WUA*

## 2D Database

The existing database is named Phabsim2DData.mdb and is located in the 2D Data folder. This database can be used to add new data to the 2D Viewer. Any data added to this database will be appended to the bottom of the database, following the existing data. To use a new database, copy the Blank.mdb database from the 2D Data folder and rename it with the desired name.

## Fill in initial data in the database (this must be done before using the importer)

- Input the Station information in tblStations:

*Station_Name* : The name of the Station you are working on

*Flow_Direction* : The flow direction of the water. The possible values are *NW* (North West), *N* (North), *NE* (North East), *E* (East), *SE* (South East), *S* (South), *SW* (South West), or *W* (West). Just use the 1 or 2 letter value.

- For each station, input the follow data into the database:
  - Flow info into tblFlows (do one row for each flow value)

    *Station_ID* : The value from the *Station_ID* field in tblStations that corresponds with the station that you are currently working with.

    *Flow_cms* : This is the flow value in ($m^3$/s) or cubic meters per second

    *Flow_cfs* : This is the flow value in ($ft^3$/s) or cubic feet per second

    *AVI_frame* : This is the corresponding frame number for the flow value in the AVI files for the given station

  - Species info into tblSpecies (do one row for each species)

    *Station_ID* : The value from the *Station_ID* field in tblStations that corresponds with the station that you are currently working with.

    *Species_Name* : The name of the fish species

## Import Mesh Data

See the following section titled "Using the 2D Importer" for screen shots and further detail on performing these steps.

Using the Importer, import the following kinds of data:

- Habitat
- Flow
- Species

## Input the 2D Data that is for All Stations

These steps have been completed for existing data, to add new data that uses different HSC Curve data and/or fish distribution data, a new database must be used.

***HSC Curves data***

- Copy the HSC Master Curves file into the HSC Folder (see folder structure above)
- Input the HSC Curves data into the database. Make an entry in the tblMediaFiles table. The data to be entered for the database fields are:

*Station_ID* : input the value = 0 (zero) as it is not associated with any particular station. (It is associated with all stations)

*Media_Type_ID* : input the value = 0 (zero) as it is not associated with any of the designated Miscellaneous Data types

*Media_Filename* : the location\filename of the HSC Master Curves file relative to the database. (i.e. If the database is located in *c:\2D Data\database.mdb* and the HSC Master Curves file is located in *c:\2D Data\ HSC\mastercurvesfilename.**, then the entry = *HSC\mastercurvesfilename.**)

*Comments* : Put in a comment saying something like "This is the Habitat Suitability Curves (HSC) definitions file -> it is not dependent on site, but just needed someplace to store the filename, so put it here."

***Fish Distribution data***

- Copy the Background Layers (hillshade image and other background shapefile layers) into the Background Layers folder located inside the Fish Distribution folder (see folder structure above)
- Input the Background Layers information into the database. Make an entry for each Background Layer in the tblFishDistribution table. The order that the background layers are entered matters, as the top most layer will be added first to the view, the next one second, etc. So, the top-most layer in the database will be the bottom-most layer in the view. The background layers need to be in this order:

  - Hillshade
  - Marine Boundary
  - Watershed Boundary
  - County Boundary

The data to be entered for the database fields are:

*Species_Name : For all Background layers enter the value = ALL. (Since the background layers are for all species).*

*FD_FileType : The type of background layer that it is. (i.e. If it is a Hillshade, then the entry = Hillshade. If it is a Watershed Boundary, then the entry = Watershed Boundary).*

*FD_Filename : The location\filename of the background layer that you are working on, relative to the database. (i.e. if the database is located*

*in c:\2D Data\database.mdb and the background layer that you are working on is located in c:\2D Data\Fish Distribution\Background Layers\layerfilename.shp then the entry = Fish Distribution\Background Layers\layerfilename.shp).*

- Copy the Fish Distribution shapefile layers into the Distribution Layers folder located inside the Fish Distribution folder (see folder structure above)
- Input the Fish Distribution Layers information into the database.
- Make an entry for each Fish Distribution Layer in the tblFishDistribution table. The order of these files does NOT matter. The data to be entered for the database fields are:

*Species_Name* : The name of the fish species that the distribution shapefile layer is for

*FD_FileType* : For all Fish Distribution layers, enter the value = Fish Distribution (as that is the type of layer that it is)

*FD_Filename* : The location\filename of the Fish Distribution layer that you are working on, relative to the database. (i.e. if the database is located in *c:\2D Data\database.mdb* and the Fish Distribution layer that you are working on is located in *c:\2D Data\Fish Distribution\Distribution Layers\layerfilename.shp* then the entry = *Fish Distribution\Distribution Layers\layerfilename.shp*).

## Input the 2D Data that is Station specific

Do each of these kinds of data for each Station:

***Background Images (must be of type *.bmp, *.gif, or *.png)***

- Copy the Background Image for the mesh into the Background Image folder (see folder structure above)
- Input the Background Image information into the database. Make an entry in the tblBackgroundImageFiles table. The data to be entered for the database fields are:

*Station_ID : The value from the Station_ID field in tblStations that corresponds with the station that you are currently working with.*

*Image_Filename : The location\filename of the Background Image, relative to the database. (i.e. If the database is located in c:\2D Data\database.mdb and the Background Image is located in c:\2D Data\StationFolderName\Background Image\imagefilename.* then the entry = StationFolderName\Background Image\imagefilename.*).*

***Fish Observation Overlays***

- Copy each of the Fish Observation Overlay shapefiles into the FishObs Overlays folder (see folder structure above).
- Input the Fish Observation Overlay layers information into the database. Make an entry for each Fish Observation Overlay layer in the tblFishObsFiles table. The data to be entered for the database fields are:

*Station_ID :* *The value from the Station_ID field in tblStations that corresponds with the station that you are currently working with.*

*FishObs_Filename : The location\filename of the Fish Observation Overlay layer that you are working on, relative to the database. (i.e. if the database is located in c:\2D Data\database.mdb and the Fish Observation Overlay layer that you are working on is located in c:\2D Data\StationFolderName\FishObs Overlays\overlayfilename.shp then the entry = StationFolderName\FishObsOverlays\overlayfilename.shp).*

***Miscellaneous data***

For each of the following Miscellaneous Data Types that you have, fill in the data.

*Aerial View*

- Copy the Aerial View image(s) into the Aerial View folder in the Miscellaneous folder (see folder structure above)
- Input the Aerial View information into the database. Make an entry for each Aerial View Image in the tblMediaFiles table. The data to be entered for the database fields are:

*Station_ID* : The value from the *Station_ID* field in tblStations that corresponds with the station that you are currently working with.

*MediaType_ID* : Select the value = *Aerial View* from the drop down combo-box in this field (It is tied to the tblMediaTypes table for values).

*Media_Filename* : The location\ filename of the Aerial View Image that you are working on, relative to the database. (i.e. if the database is located in *c:\2D Data\database.mdb* and the Aerial View Image that you are working on is located in *c:\2D Data*\StationFolderName*Miscellaneous*\ *Aerial View**imagefname.** then the entry = StationFolder Name*Miscellaneous**Aerial View**imagefname.**).

*Comments* : Put any comments about the Aerial View image you are working on that you want a user to know here. This value will be displayed when this Aerial View image is being viewed. If this field is left blank, then the value **None** will be displayed instead.

**AVI**

- Copy the AVIs into the AVI folder in the Miscellaneous folder (see folder structure above)
- Input the AVI information into the database. Make an entry for each AVI in the tblMediaFiles table. The data to be entered for the database fields are:

*Station_ID* : The value from the *Station_ID* field in tblStations that corresponds with the station that you are currently working with.

*MediaType_ID* : Select the value = *AVI* from the drop down combo-box in this field (It is tied to the tblMediaTypes table for values).

*Media_Filename* : The location\ filename of the AVI that you are working on, relative to the database. (i.e. if the database is located in *c:\2D Data\database.mdb* and the AVI that you are working on is located in *c:\2D Data*\StationFolderName\ *Miscellaneous*\ *AVI\avifilename.avi* then the entry = StationFoldername\ *Miscellaneous\AVI\avifilename.avi*).

*Comments* : Put any comments about the AVI you are working on that you want a user to know here. This value will be displayed when this AVI is being viewed. If this field is left blank, then the value **None** will be displayed instead.

*PDF*

- Copy the PDF documents into the PDF folder in the Miscellaneous folder (see folder structure above)
- Input the PDF documents information into the database. Make an entry for each PDF document in the tblMediaFiles table. The data to be entered for the database fields are:

*Station_ID* : The value from the *Station_ID* field in tblStations that corresponds with the station that you are currently working with.

*MediaType_ID* : Select the value = *PDF* from the drop down combo-box in this field (It is tied to the tblMediaTypes table for values).

*Media_Filename* : The location\ filename of the PDF that you are working on, relative to the database. (i.e. if the database is located in *c:\2D Data\database.mdb* and the PDF that you are working on is located in *c:\2D Data*\StationFolder Name*Miscellaneous*\ *PDF\pdffilename.pdf* then the entry = StationFolderName\ *Miscellaneous\PDF*\ *pdffilename.pdf*).

*Comments* : Put any comments about the PDF document you are working on that you want a user to know here. This value

will be displayed when this PDF document is being viewed. If this field is left blank, then the value **None** will be displayed instead.

**Photos**

- Copy the Photographs into the Photos folder in the Miscellaneous folder (see folder structure above)
- Input the Photographs information into the database. Make an entry for each Photograph in the tblMediaFiles table. The data to be entered for the database fields are:

*Station_ID* : The value from the *Station_ID* field in tblStations that corresponds with the station that you are currently working with.

*MediaType_ID* : Select the value = *Photograph* from the drop down combo-box in that field (It is tied to the tblMediaTypes table for values).

*Media_Filename* : The location\ filename of the Photograph that you are working on, relative to the database. (i.e. if the database is located in *c:\2D Data\database.mdb* and the Photograph that you are working on is located in *c:\2D Data*\StationFolderName*Miscellaneous\Photos\photofilename.** then the entry = StationFolderName*Miscellaneous\Photos\photofilename.**).

*Comments* : Put any comments about the Photograph you are working on that you want a user to know here. This value will be displayed when this Photograph is being viewed. If this field is left blank, then the value **None** will be displayed instead.

***WUA Data***

- Copy the WUA file into the WUA folder (see folder structure above)
- Input the WUA information into the database. Make an entry in the tblWUAFiles table. The data to be entered for the database fields are:

*Station_ID : The value from the Station_ID field in tblStations that corresponds with the station that you are currently working with.*

*WUA_Filename : The location\filename of the WUA file, relative to the database. (i.e. if the database is located in c:\2D Data\ database.mdb and the WUA file is located in c:\2D Data\ StationFolderName\WUA\wuafilename.* then the entry = StationFolderName\WUA\wuafilename.*).*

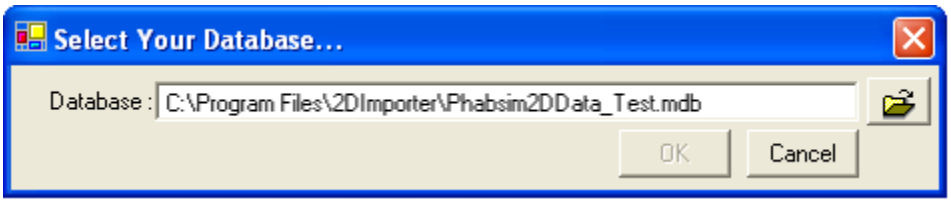
# Appendix B-3: Using the 2D Data Importer

## Install 2D Data Importer

To install the 2D Data Importer, double click on the setup.exe file. This will install the importer as well as these help files and a blank database. The default location for the installation of these items is C:\Program Files\2DImporter and C:\2D_Importer_Output.

## Select Database

Launch the 2D Data Importer program. The default location for the installation of this program is C:\Program Files\2DImporter. Find the executable named 2DMeshDataImporter.exe. Double click this application to begin. The following screen will appear:

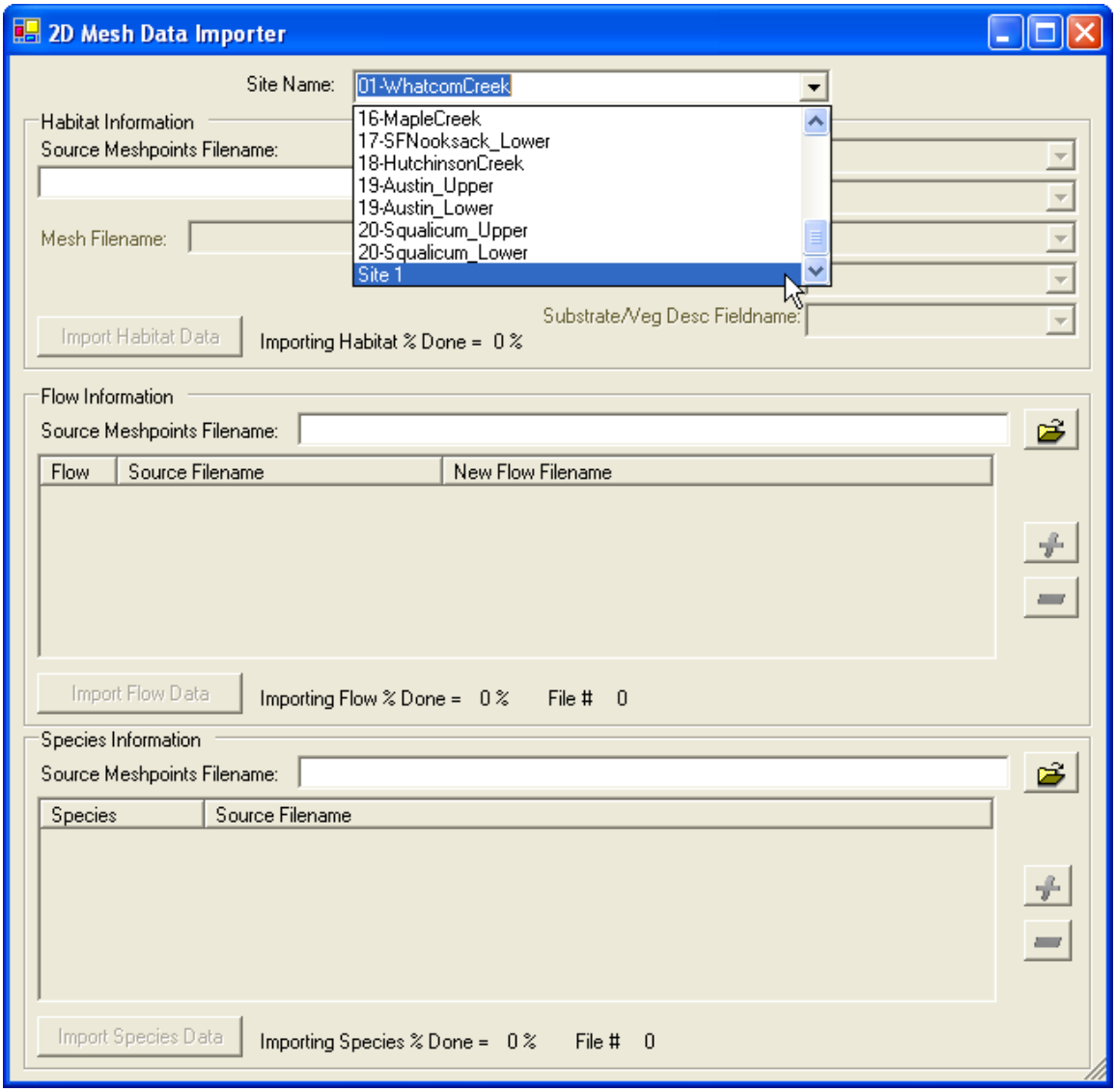



Select the desired database. In the example above, this is C:\Program Files\2DImporter\ PHabSim2DData_Test.mdb.

To append data to the database used by the 2D Viewer, select the database named Phabsim2DData.mdb. The default location for this file is C:\Program Files\WRIA-1_DSS\DSS_Data\2D Data\ Phabsim2DData.mdb. If the DSS was not installed in the default location, this file will have to be located.

To start a new, blank database, copy the Blank.mdb database from the 2D Data folder, rename it, and enter its location in the box above.

After selecting the database, the main 2D Data Importer window below will appear. Select the desired site name from the drop-down list.



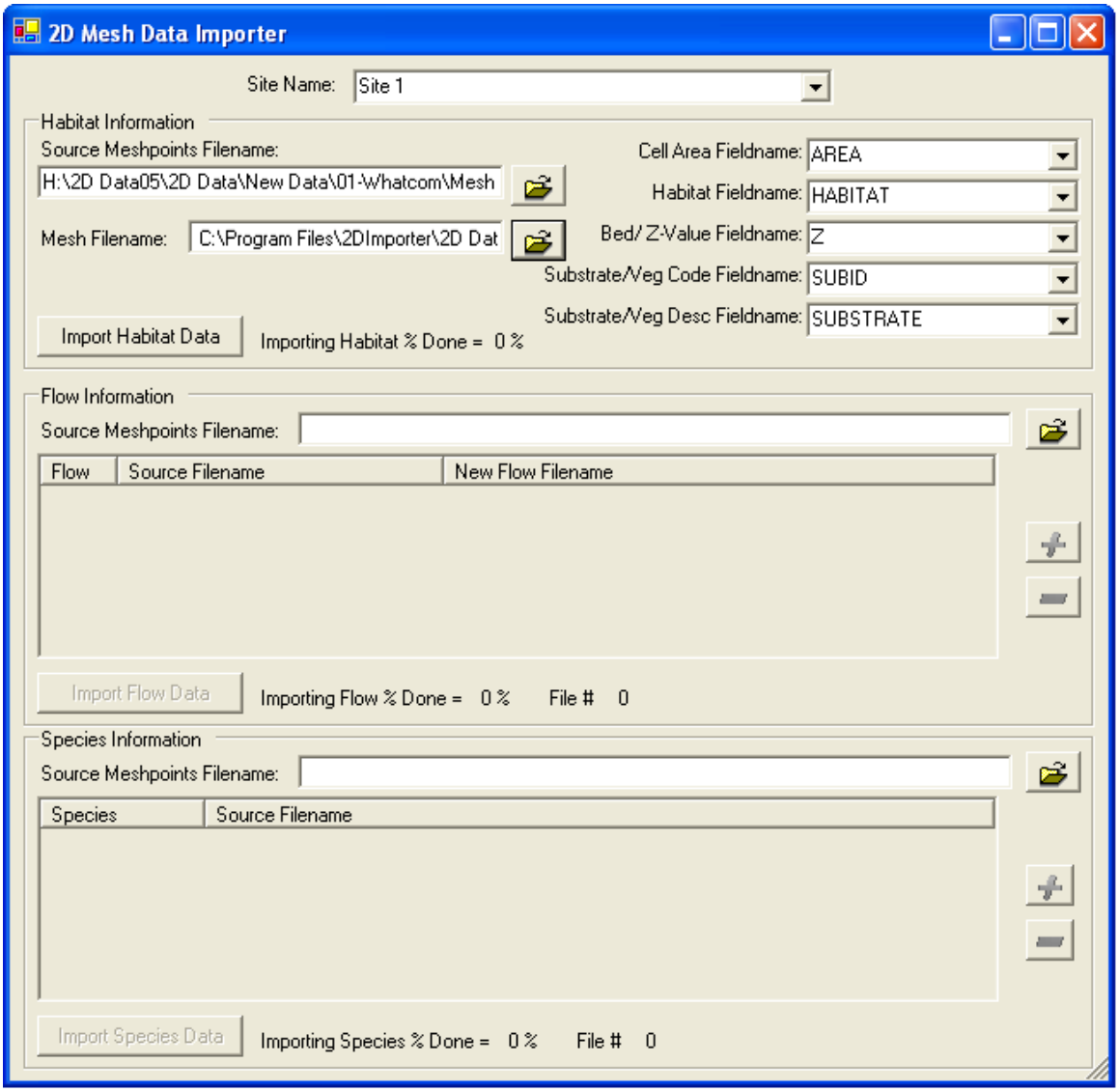

Note that only sites contained in the database just selected will be available. Before using a new, blank database this information must be entered as explained in the previous section titled "How to Import 2D Data".

## Importing Habitat Data

To begin importing habitat data, either specify the data path, or push the Browse button next to the "Source Meshpoints Filename:" textbox to select your source point shapefile. Once a valid file has been selected, the drop-down lists at the right, and the Mesh Filename textbox and Browse button will become active.

Next, select the appropriate field names from the drop-down lists at the right. Finally, specify the filename for the output mesh file. The file may be named as desired, but it must have a *.shp* extension. To select a specific location, use the Browse button next to

the “Mesh Filename:” textbox. Once these two steps have been properly performed, the Import Habitat Data button will be enabled and the screen will look similar to the screen shown below:

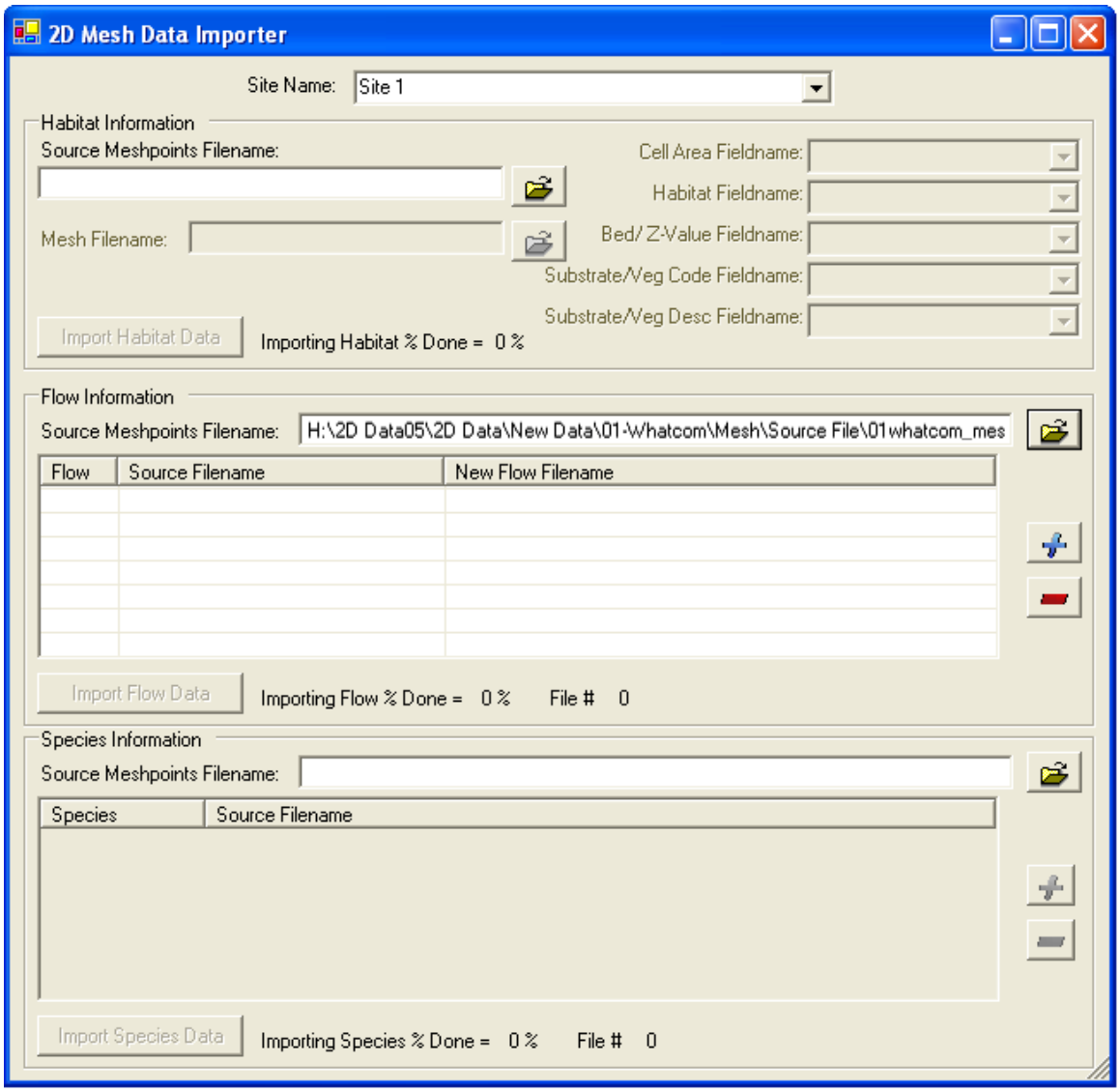



To import the data, click the “Import Habitat Data” button. A progress update may appear, but it is more likely that the application will become unresponsive for a few minutes while this process is completed due to the complexity of the operation being performed.

When the program returns to operation, the file will have been created and an entry created for it in the database.

## Importing Flow Data

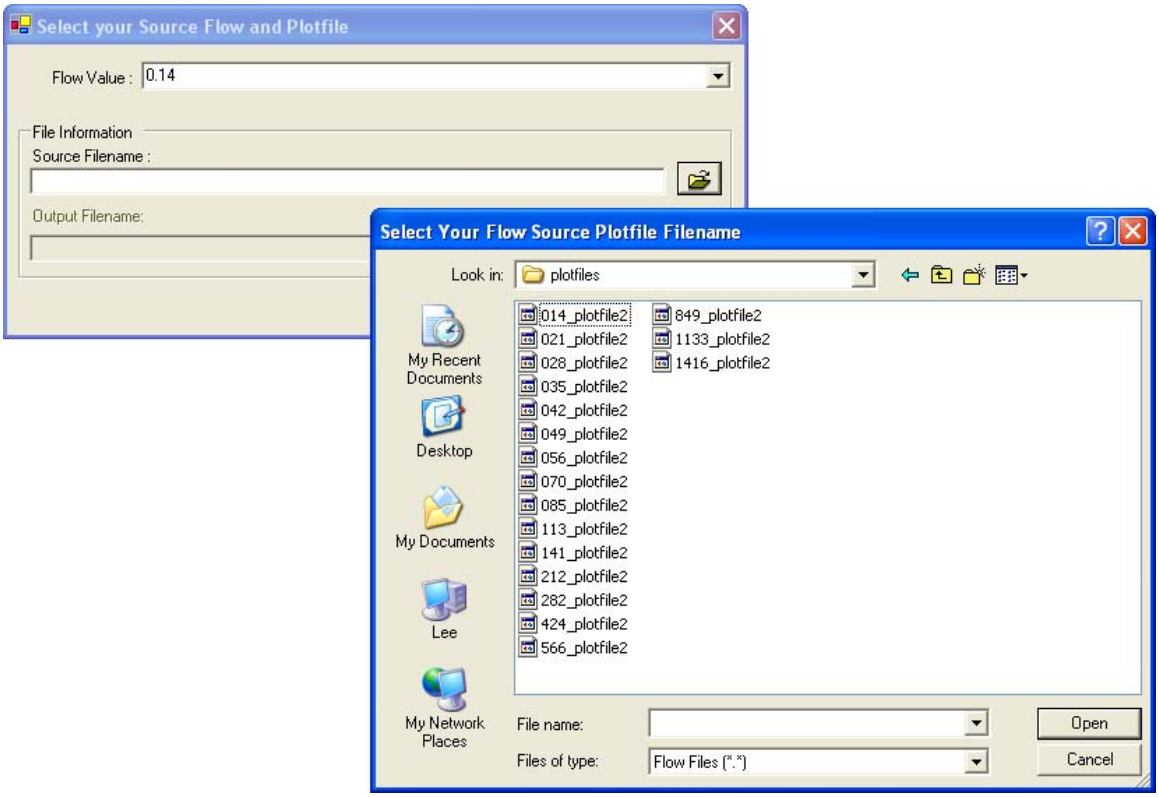
To import Flow data, click the Browse button next to the "Source Meshpoints Filename:" to select the appropriate source shapefile. This is the same shapefile used for the Source Meshpoints Filename when importing Habitat data. When this is done, the File list and Add/Remove File buttons will be enabled. The source mesh file is shown already selected in the screen below.



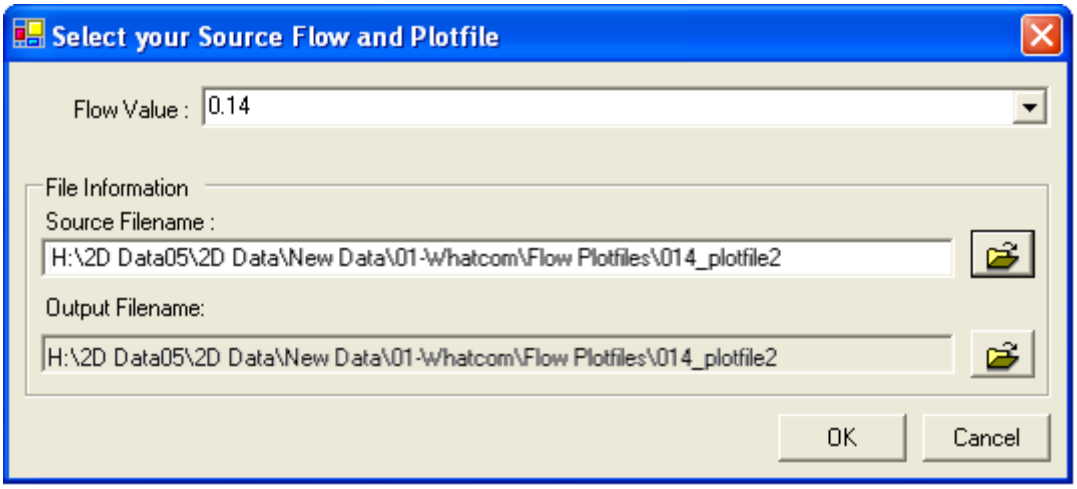

### *Adding Flow Source Plotfiles*

To add the Flow Source Plotfiles, click the Plus icon on the right of the File list. The "Select your Source Flow and Plotfile" screen will appear as below. Choose the

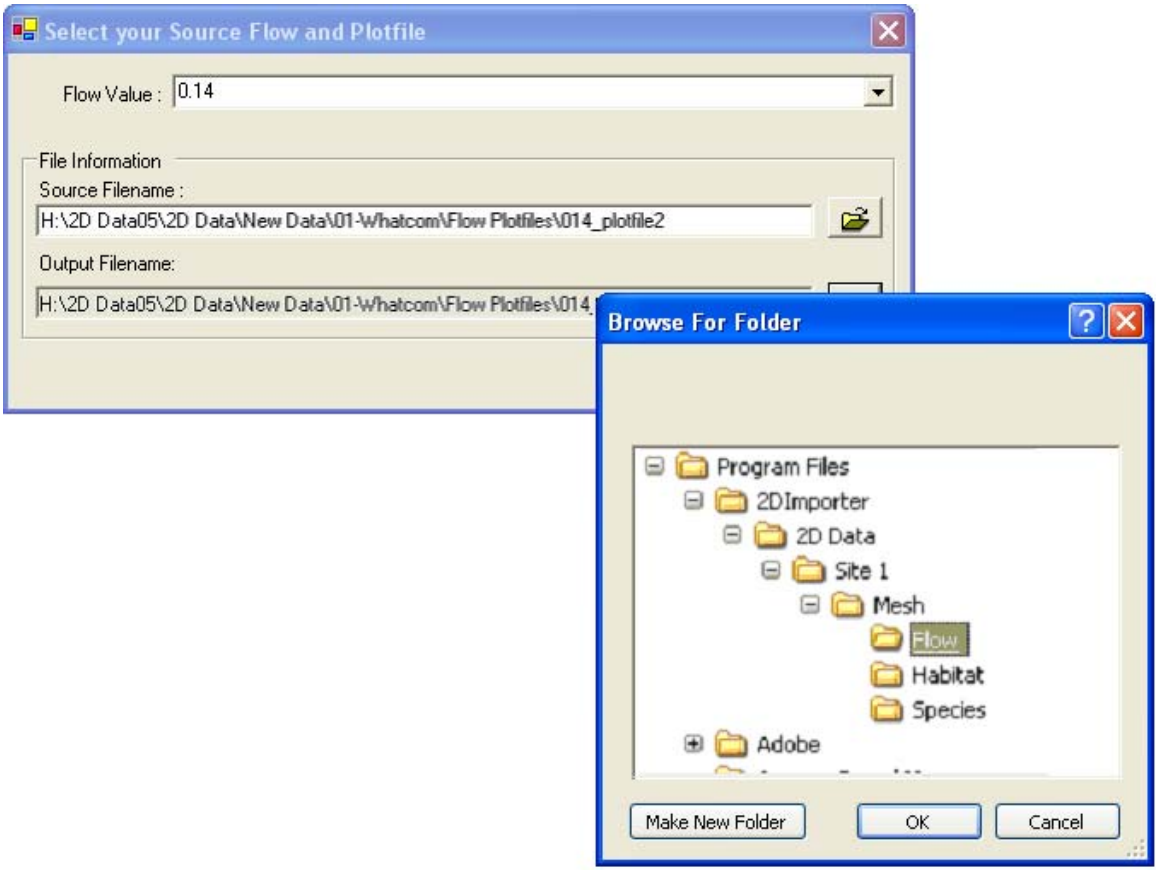
appropriate flow value, and then click Browse to locate the source plotfile. The "Select Your Flow Source Plotfile Filename" window will appear as shown below.



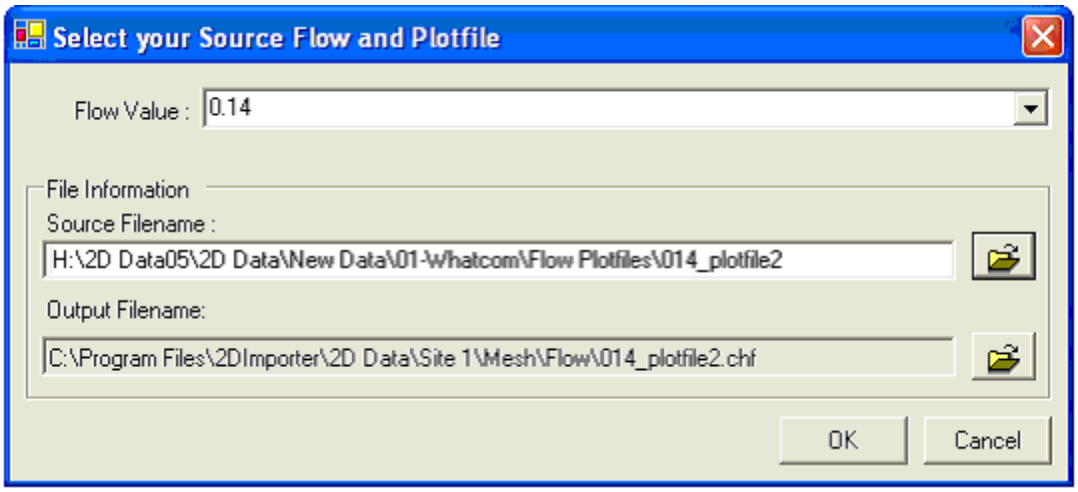

Select the appropriate file and the window will be completely filled in as shown below.



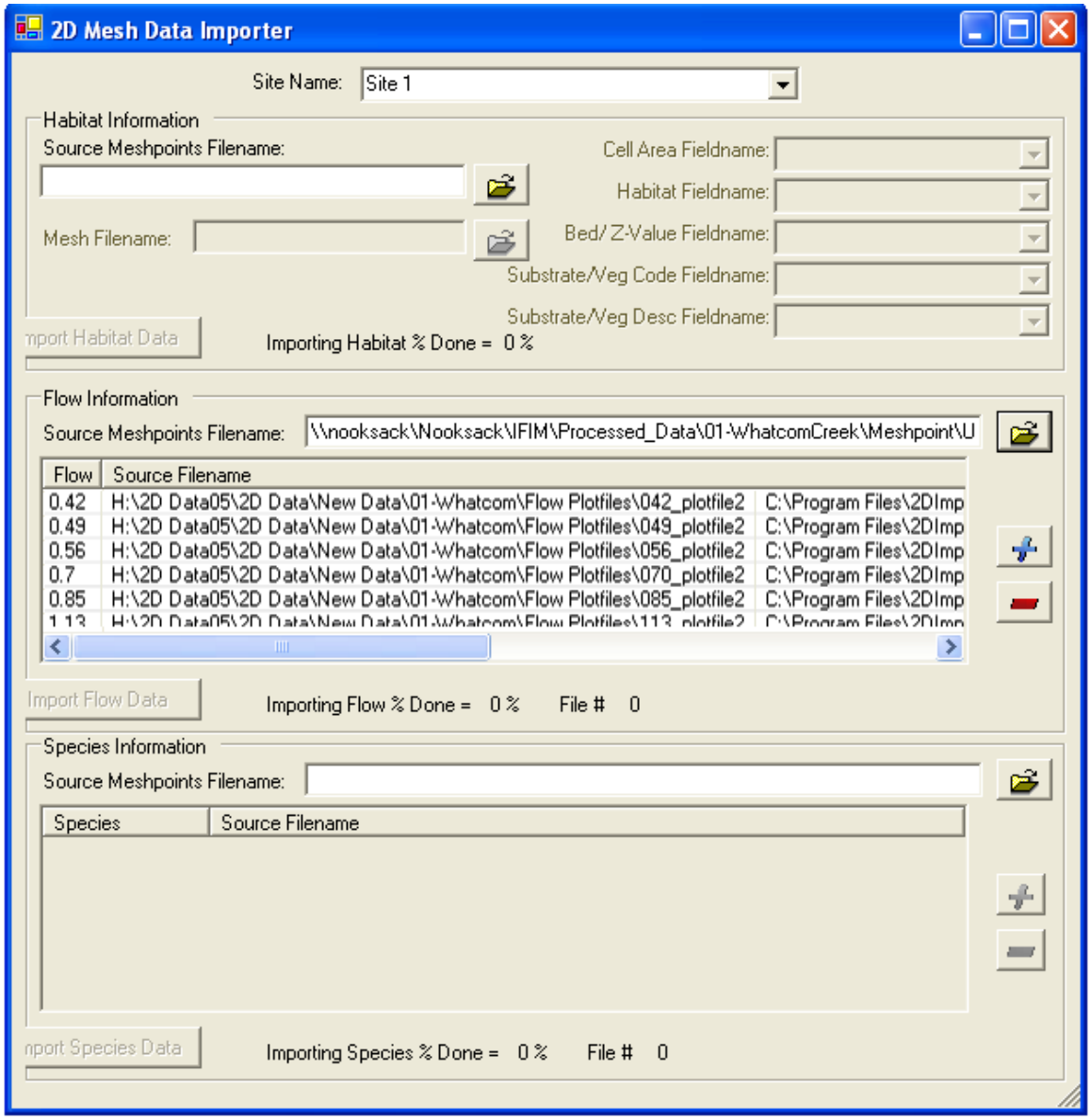

To change the directory where the output files will be written, click the second browse button by Output Filename. A form titled "Browse For Folder" will appear as below.




The Output Filename value will be changed to reflect the newly selected directory. It will now look like the form below.




Repeat this process until there is an entry to the File list for every flow value. At this point, the screen will look as it does below:

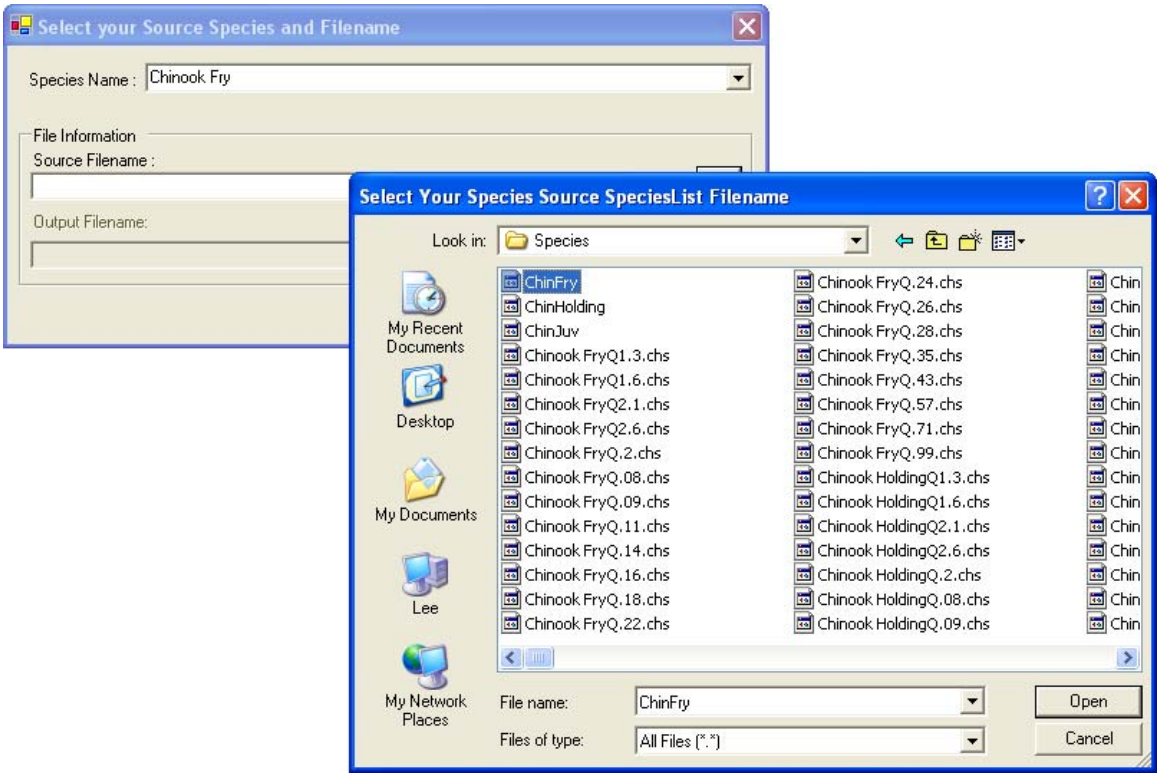

2D Mesh Data Importer
Site Name: Site 1
Habitat Information
Source Meshpoints Filename:
Cell Area Fieldname:
Habitat Fieldname:
Mesh Filename:
Bed/ Z-Value Fieldname:
Substrate/Veg Code Fieldname:
Substrate/Veg Desc Fieldname:
nport Habitat Data
Importing Habitat % Done = 0 %
Flow Information
Source Meshpoints Filename: \\nooksack\Nooksack\IFIM\Processed_Data\01-WhatcomCreek\Meshpoint\U
Flow
Source Filename
0.42 H:\2D Data05\2D Data\New Data\01-Whatcom\Flow Plotfiles\042_plotfile2 C:\Program Files\2DImp
0.49 H:\2D Data05\2D Data\New Data\01-Whatcom\Flow Plotfiles\049_plotfile2 C:\Program Files\2DImp
0.56 H:\2D Data05\2D Data\New Data\01-Whatcom\Flow Plotfiles\056_plotfile2 C:\Program Files\2DImp
0.7 H:\2D Data05\2D Data\New Data\01-Whatcom\Flow Plotfiles\070_plotfile2 C:\Program Files\2DImp
0.85 H:\2D Data05\2D Data\New Data\01-Whatcom\Flow Plotfiles\085_plotfile2 C:\Program Files\2DImp
Import Flow Data
Importing Flow % Done = 0 % File # 0
Species Information
Source Meshpoints Filename:
Species
Source Filename
nport Species Data
Importing Species % Done = 0 % File # 0

To import the flow data, click the Import Flow Data button. The progress will be displayed as the data is imported. Once this finishes, each of the output files (*.chf) will be written to the output locations listed in the File list, and an entry put into the database for each of them.

## Importing Species Data

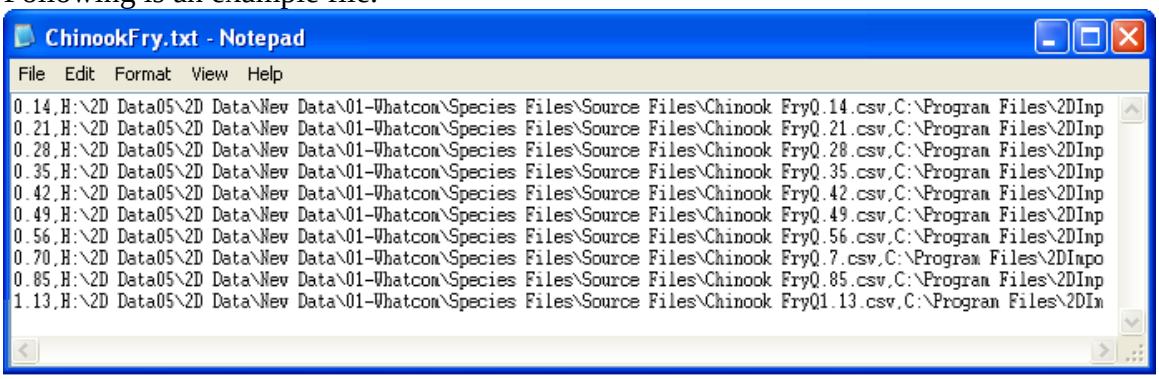
Importing the Species data is very similar to importing the Flow data. To begin, click the Browse button next to the “Source Meshpoints Filename:” and select the appropriate source shapefile. Again, this is the same source mesh file that was used when importing the Habitat and/or the Flow data. Also, as when importing the Flow data, the File list and Add/Remove buttons will enabled once the source mesh file has been selected.

Next, add the Species and the Species List source files. To do this, click the Plus icon to bring up the "Select your Source Species and Filename" screen below. Choose the appropriate Species Name then click Browse to locate the source Species List file. The "Select Your Species Source SpeciesList Filename" window appears as shown below.

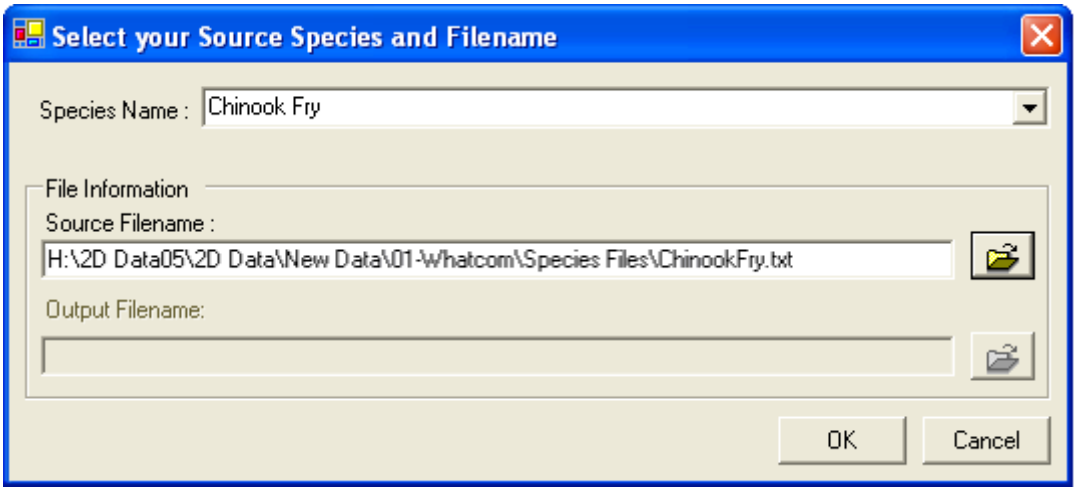



The SpeciesList file should be in a specific format:
*FlowValue, Source Filename (*.csv), Output Filename (*.chs)*

Following is an example file.

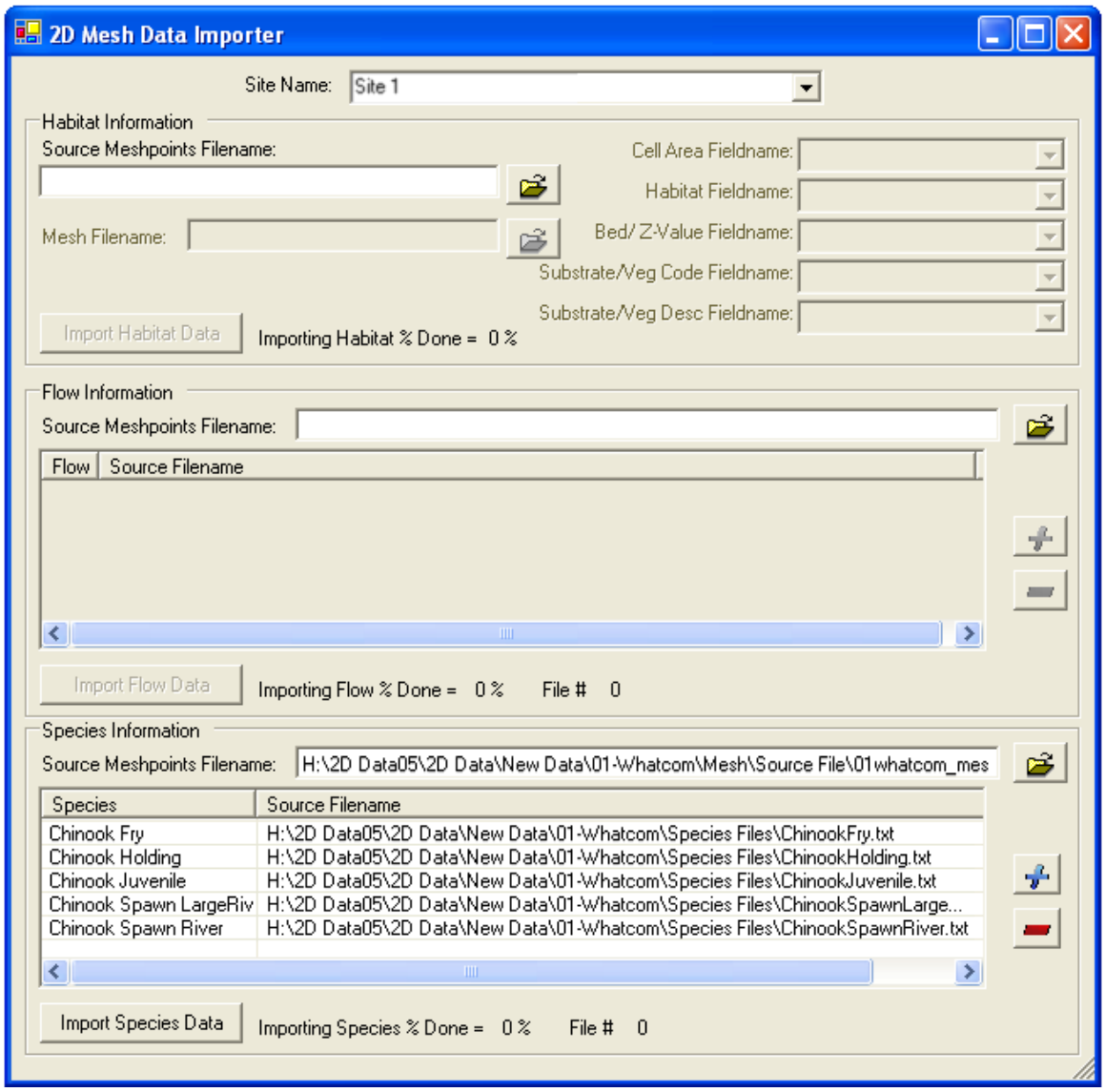

ChinookFry.txt - Notepad
File Edit Format View Help
0.14,H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\Source Files\Chinook FryQ.14.csv,C:\Program Files\2DInp
0.21,H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\Source Files\Chinook FryQ.21.csv,C:\Program Files\2DInp
0.28,H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\Source Files\Chinook FryQ.28.csv,C:\Program Files\2DInp
0.35,H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\Source Files\Chinook FryQ.35.csv,C:\Program Files\2DInp
0.42,H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\Source Files\Chinook FryQ.42.csv,C:\Program Files\2DInp
0.49,H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\Source Files\Chinook FryQ.49.csv,C:\Program Files\2DInp
0.56,H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\Source Files\Chinook FryQ.56.csv,C:\Program Files\2DInp
0.70,H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\Source Files\Chinook FryQ.7.csv,C:\Program Files\2DInpo
0.85,H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\Source Files\Chinook FryQ.85.csv,C:\Program Files\2DInp
1.13,H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\Source Files\Chinook FryQ1.13.csv,C:\Program Files\2DIn

After choosing the appropriate file, the window will be completely filled in as shown below.




Because the SpeciesList file lists all of the source and output files for the selected species, the Output Filename value is not used and may be ignored.

Repeat this process of selecting Species/SpeciesList pairs until an entry has been added to the File list for every species. At this point, the screen will look as it does below:


2D Mesh Data Importer
Site Name: Site 1
Habitat Information
Source Meshpoints Filename:
Mesh Filename:
Cell Area Fieldname:
Habitat Fieldname:
Bed/ Z-Value Fieldname:
Substrate/Veg Code Fieldname:
Substrate/Veg Desc Fieldname:
Import Habitat Data
Importing Habitat % Done = 0 %
Flow Information
Source Meshpoints Filename:
Flow
Source Filename
Import Flow Data
Importing Flow % Done = 0 %
File # 0
Species Information
Source Meshpoints Filename: H:\2D Data05\2D Data\New Data\01-Whatcom\Mesh\Source File\01whatcom_mes
Species
Source Filename
Chinook Fry H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\ChinookFry.txt
Chinook Holding H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\ChinookHolding.txt
Chinook Juvenile H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\ChinookJuvenile.txt
Chinook Spawn LargeRiv H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\ChinookSpawnLarge...
Chinook Spawn River H:\2D Data05\2D Data\New Data\01-Whatcom\Species Files\ChinookSpawnRiver.txt
Import Species Data
Importing Species % Done = 0 %
File # 0

Click the Import Species Data button, and the progress will be displayed as the data is imported. Once this finishes, each of the output files (*.chs) will be written to the output locations listed from the Source Filename listed in the File list, and an entry put into the database for each of them.

Mesh, flow and species data should now be written to the selected database for the selected site. To import data for another site, select a new site name from the list and repeat the importing steps.

6007

# Appendix C:
# Extrapolation Program User Manual

## Extrapolation Program

### Introduction

The program extrapolates instream flow recommendations (IFR) and/or weighted usable area (WUA) curves from 37 study sites (20 intensive IFIM sites and 17 rapid assessment sites) to the additional 300 nodes within the Nooksack watershed.

General guidelines when extrapolating IFR or WUA curves:

1. Choose a USU intensive study site over a Rapid Assessment study site;
2. Choose a USU study site located geographically closer to the site of interest (e.g., Hutchinson USU intensive study site should be used for extrapolation within subcluster 3 (Hydro Cluster 7); however, Deer Creek rapid assessment site should be used for extrapolation within drainage 180); or
3. Choose a USU study site with MAF closer to that of the site of interest;
4. Caution should be exercised when extrapolating IFRs or WUA curves from the study sites that were not modeled over the entire range of flows (10% to 90% exceedence flows) due to modeling constraints (Maple Creek, Kendall Creek, Lynden, Bell Creek, Canyon Lake Creek, Lower Ruth Creek, Porter Creek), since no detailed information is available on the behavior of the curves at higher flows.

A more detailed description of the method can be found in Saraeva (2006; http://www.engineering.usu.edu/uwrl/inse/nooksack/Saraeva/Dissertation.pdf), chapter 2 (p.9).

### Program Setup

The program asks for an input of the "Source – Target" file. The file should be comma delimited and should have 8 columns: Node, DrainId, Cluster, Subcluster, MAF_cfs, 60%exeed_cfs, Source _Site, Site_Type and 338 rows (337 node records and a header). The file provided with the program is "Source-Target_337nodes.csv". Figure 1 shows the first few records of the file.

Intensive IFIM study sites should have "Intensive Site" in the Site_Type column; rapid assessment study sites should have "1D Site" in the Site_Type column.

```
Node, DrainId,Cluster,Subcluster,MAF_cfs,60%exeed_cfs,Source _Site,Site_Type
166,79,8,6,23.8,7.9,AndersonCreek,Drainage Outlet
167,79,8,6,21.4,7.2,AndersonCreek,Streamflow Gage
168,79,8,6,17.1,5.8,AndersonCreek,Streamflow Gage
169,79,8,6,15.7,5.4,AndersonCreek,Intensive Site
256,56,8,6,22.8,7.7,AndersonCreek,Drainage Outlet
272,200,8,6,3.4,1.1,AndersonCreek,Drainage Outlet
290,107,8,3,3.2,1.2,AustinCreek,Drainage Outlet
291,110,8,3,2.1,0.9,AustinCreek,Drainage Outlet
```

Figure 1. First 8 records of the "Source-Target_337nodes.csv" file.

Four folders must be in the same directory as the executable program file:

1. Historic_337exceedences
2. WUAinput
3. IFRinput
4. Pictures

### *"Historic_337exceedences" Folder*

"Historic_337exceedences" folder contains 337 comma delimited text files with exceedence tables for the 337 nodes. Figure 2 shows an example of the file for the node 1. The folder with all the files is provided with the program.

```
Node  1
Exceedance,Oct,Nov,Dec,Jan,Feb,Mar,Apr,May,Jun,Jul,Aug,Sep
5,7533.86,11520.32,11800.43,10378.75,9451.05,7508.35,6595.58,7096.31,7339.58,5826.44,3642.87,3988.47
10,5480.22,9320.56,9179.83,8322.43,7822.86,6200.5,5727.69,6357.36,6536.46,5146.62,3051.4,2992.76
15,4623.53,7981.16,7985.3,7303.16,6937,5432.93,5310.27,5938.31,6085.79,4650.59,2788.13,2586.61
20,3907.4,7228.76,6989.76,6599.82,6069.37,4895.83,5013.46,5573.24,5786.49,4160.54,2513.63,2238.82
25,3430.16,6548.87,6300.62,5972.55,5425.72,4488.7,4719.43,5344.09,5468.49,3833.58,2346.92,2020.03
30,3048.68,5947.3,5824.86,5484.39,4890.23,4209.61,4495,5116.65,5227.54,3606.89,2218.7,1846.37
35,2725.71,5432.01,5401.55,5051.96,4464.67,3944.68,4279.28,4874.02,4944.08,3389.33,2110.82,1748.96
40,2411.62,5045.17,5068.18,4695.3,4121.19,3760.58,4103.26,4686.62,4687.63,3182.1,2032.54,1648.03
45,2117.68,4609.47,4749.73,4343.06,3840,3525.56,3948.37,4475.99,4472.16,3019.03,1923.07,1552.06
50,1942.1,4292.53,4423.7,3970.5,3577.88,3374.18,3783.97,4271.68,4274.83,2871.33,1846.33,1461.17
55,1738.32,3920.42,4138.88,3727.5,3338.43,3207.68,3648.44,4134.65,4084.32,2766.07,1768.9,1374.65
60,1560.29,3615.08,3870.46,3521.99,3128.88,3086.12,3510.52,4017.66,3911,2658.2,1699.53,1302.53
65,1422.41,3309.41,3586.18,3317.92,2958,2961.01,3399.3,3852.33,3686.22,2560.31,1629.2,1227.27
70,1262.98,3062.18,3319.37,3069.75,2812.17,2825.37,3274.95,3722.19,3515.74,2428.52,1564.51,1179.21
75,1149.48,2833.97,3047.93,2821.89,2660.64,2669.01,3136.01,3601.48,3387.98,2312.03,1495.24,1123.29
80,1074.3,2493.36,2770.96,2585.02,2462.78,2501.88,2969.53,3444.27,3217.13,2180.33,1425.84,1065.11
85,985.25,2184.08,2448.7,2312.45,2286.43,2307.49,2806.03,3238.13,3030.97,2047.87,1350.15,987.51
90,881.46,1796.24,2144.87,2047.87,2111.8,2106.08,2583.31,3067.75,2809.61,1920.78,1264.82,904.56
95,755.47,1401.66,1854.05,1654.61,1781.7,1840.95,2281.51,2826.09,2577.79,1786.61,1150.71,793.49
```

Figure 2. Exceedence file for node 1.

### *"WUAinput" Folder*

The "WUAinput" folder contains 37 comma delimited files with WUA data for each of the study sites. The files should have flows and WUAs for 28 fish species and life stages in the same format as in the deliverable WUA excel files for each site. Example of the file is shown in figure 3. The first line indicates the number of modeled flows. Note that the names of the files, e.g., AndersonCreek (.csv), AustinCreek (.csv), should be exactly the same as in the "Source_Site" column of the Source-Target file (Figure 1). The folder is provided with the program.

```
16
Chinook Spawn Stream,,Chinook Spawn River,,Chinook Spawn Large River,,Chinook Holding,
2.83,163.77,2.83,80.15,2.83,88.48,2.83,2932.4,2.83,271.23,2.83,5797.42,2.83,5457.8,2.8
4.94,1427.8,4.94,724.52,4.94,785.37,4.94,3142.3,4.94,362.21,4.94,6103.76,4.94,6197.09,
9.89,3773.48,9.89,2357.67,9.89,2464.5,9.89,3576.63,9.89,631.73,9.89,6105.35,9.89,7446.
14.83,5593.49,14.83,4007.89,14.83,4137.41,14.83,3967.69,14.83,901.18,14.83,5789.28,14.
20.13,6999.91,20.13,5473.67,20.13,5602.56,20.13,4291.32,20.13,1144.78,20.13,5331.12,20
25.07,8030.15,25.07,6584.97,25.07,6686.81,25.07,4600.58,25.07,1393.71,25.07,5135.59,25
30.02,8821.04,30.02,7552.04,30.02,7622.5,30.02,4853.09,30.02,1535.74,30.02,5059.54,30.
34.96,9491.07,34.96,8416.11,34.96,8481.65,34.96,5056.13,34.96,1690.2,34.96,4869.49,34.
39.91,9990.36,39.91,9116.04,39.91,9183,39.91,5211.32,39.91,1769.51,39.91,4598.95,39.91
44.85,10378.92,44.85,9771.38,44.85,9841.98,44.85,5336.79,44.85,1808.97,44.85,4196.89,4
50.15,10601.78,50.15,10244.11,50.15,10335.91,50.15,5460.57,50.15,1827.66,50.15,3865.92
60.03,10912.38,60.03,10966.89,60.03,11131.69,60.03,5685.5,60.03,2101.71,60.03,3443.89,
74.87,11257.89,74.87,11748.17,74.87,12131,74.87,6076.21,74.87,2262.41,74.87,3143.67,74
99.94,10828.59,99.94,11798.31,99.94,12993.1,99.94,6612.04,99.94,2526.5,99.94,2839.1,99
125.01,9664.66,125.01,10788.86,125.01,13247.03,125.01,7072.78,125.01,2698.05,125.01,26
150.09,8022.64,150.09,9015.52,150.09,12925.98,150.09,7376.79,150.09,2965.09,150.09,253
```

Figure 3. Example of the WUA input file.

***"IFRinput" Folder***

The "IFRinput" folder contains 37 comma delimited text files with instream flow recommendations for the 37 USU study sites. Figure 4 shows an example of the IFR input file. Again, note that the names of the files, e.g., AndersonCreek (.txt), AustinCreek (.txt), should be exactly the same as in the "Source_Site" column of the Source-Target file (Figure 1). The folder is provided with the program.

```
Month,IFR_cfs
Oct,4.46
Nov,20.91
Dec,42.81
Jan,51.49
Feb,50.94
Mar,28.60
Apr,18.79
May,10.73
Jun,7.37
Jul,4.95
Aug,3.60
Sep,3.53
```

Figure 4. Example of the IFR input file.

***"Pictures" Folder***

The "Pictures" folder contains a number of pictures that are displayed during the program run. The folder is provided with the program.

## Running the program

The user may choose to extrapolate either IFR or WUA curves or both (Figure 5).

Figure 5. Extrapolation choices: IFR or WUA.

Dialog opens that asks to "Open file with Source and Target Sites". Open “Source-Target_337nodes.csv” file.

The user is asked to choose to extrapolate to all 300 nodes or a single node (Figure 6).

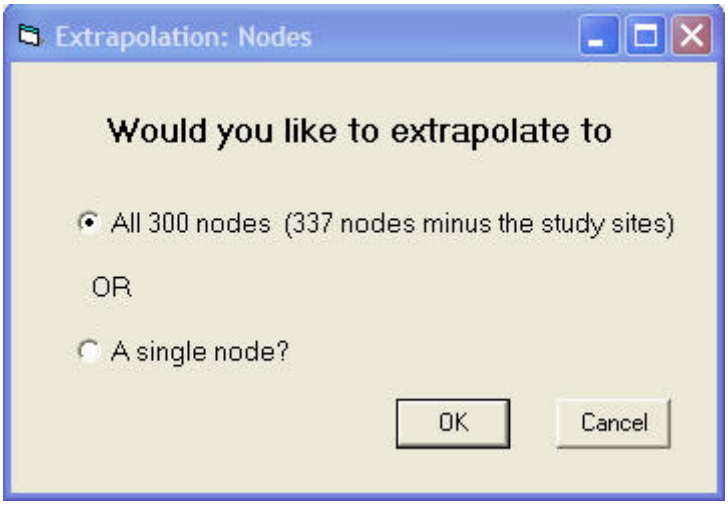



Figure 6. Extrapolation choices: 300 nodes or 1 node.

***Extrapolate to All 300 Nodes***

*Extrapolation of the IFRs*

If the user chooses the first option ("All 300 nodes") then the extrapolated IFRs will be saved into separate comma delimited files for each node in the "ExtrapolatedIFR_output" directory in the same directory where the executable program is located. If such folder already exists the user will be asked to replace it or to enter a new folder name.

*Extrapolation of the WUA curves*

The user will be given an option to save extrapolated WUA values in the individual comma delimited files or in the "tblWUA_MethodB" table of the Habitat database (Figure 7). The detailed description of the table and the database is given in the "Technical Documentation: The Habitat Time Series Model". HabitatTimeSeriesInput.mdb database is provided with the program.

If the user chooses the first option (.csv files) the extrapolated WUA will be saved into separate comma delimited files for each node in the "ExtrapolatedWUA_output" directory in the same folder where the executable program is located. If such folder already exists the user will be asked to replace it or to enter a new folder name.
If the user chooses to save the output in the Habitat database, the user will be prompted to open the database. It should be noted that even though both extrapolated WUA in sq.feet/1000ft and percent of maximum habitat are written into the database, only percent of maximum habitat values that show the shape of the WUA curves have been proven to provide meaningful results.

***Extrapolate to a Single Node***

If the user chooses to extrapolate to a single node, the next window will request the user to either enter the node number or will offer to show a map (Figure 8).

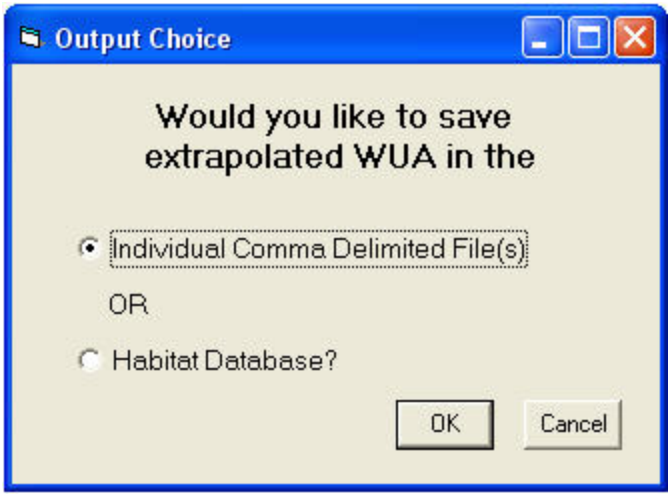



Figure 7. Output choice.

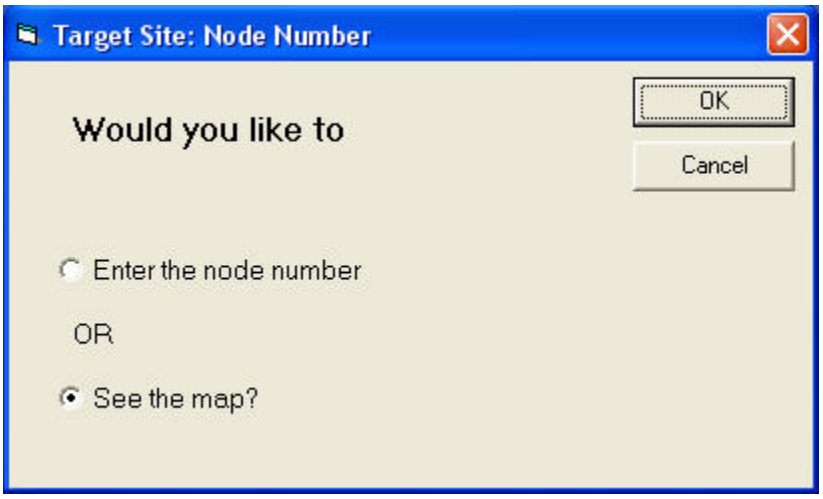



Figure 8. Choosing the target site node number.

*"Enter the Node Number" Option*

If the first option is chosen ("Enter the node number") the user will be prompted to enter the node number. If the user enters an invalid node number a message will be displayed asking to reenter the node number.

Extrapolation can only be carried out to the nodes other than USU study sites. If the entered node corresponds to one of the USU study sites the user will get a message asking to enter another node number (Figure 9).

Figure 9. The message the user gets if the entered node number is one of the USU study sites.

The user will be asked if they would like to use the default source site for extrapolation or choose a new source site (Figure 10). If the user chooses to use the default extrapolation site they will be prompted to enter the file name to save the output. If the user chooses to select another source site for extrapolation the next window will show a list of USU study sites belonging to the same hydro group as the target site (Figure 11). If the chosen target and source sites have significantly different flow magnitude (e.g., one site is a small creek and the other site is a river) the user will get a message showing flow magnitudes for both sites (Figure 12). The user has a choice to either continue with the selected source site or choose another source site for the extrapolation.

The user will then be asked to save the output file.

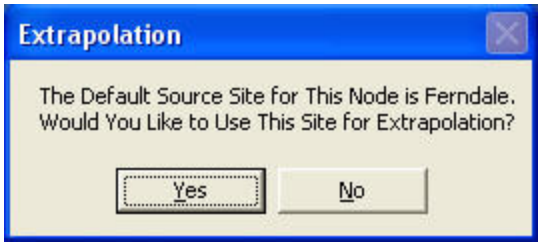

Extrapolation
The Default Source Site for This Node is Ferndale.
Would You Like to Use This Site for Extrapolation?
Yes
No

Figure 10. The message the user gets after entering the node number.

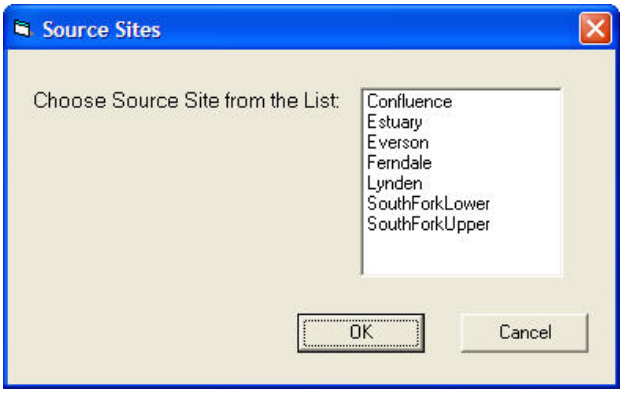



Figure 11. Example of the source site list.

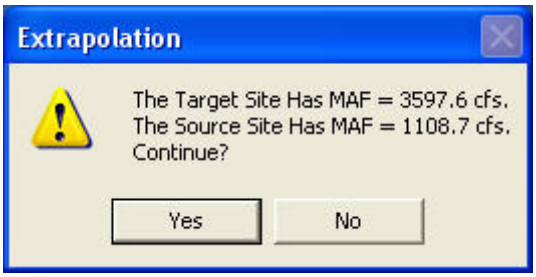



Figure 12. Message showing flow magnitudes of the source and target sites.

*"See the Map" Option*

If the user chooses to see the map (Figure 8) the next window will display the map with hydrologically similar groups (Figure 13).

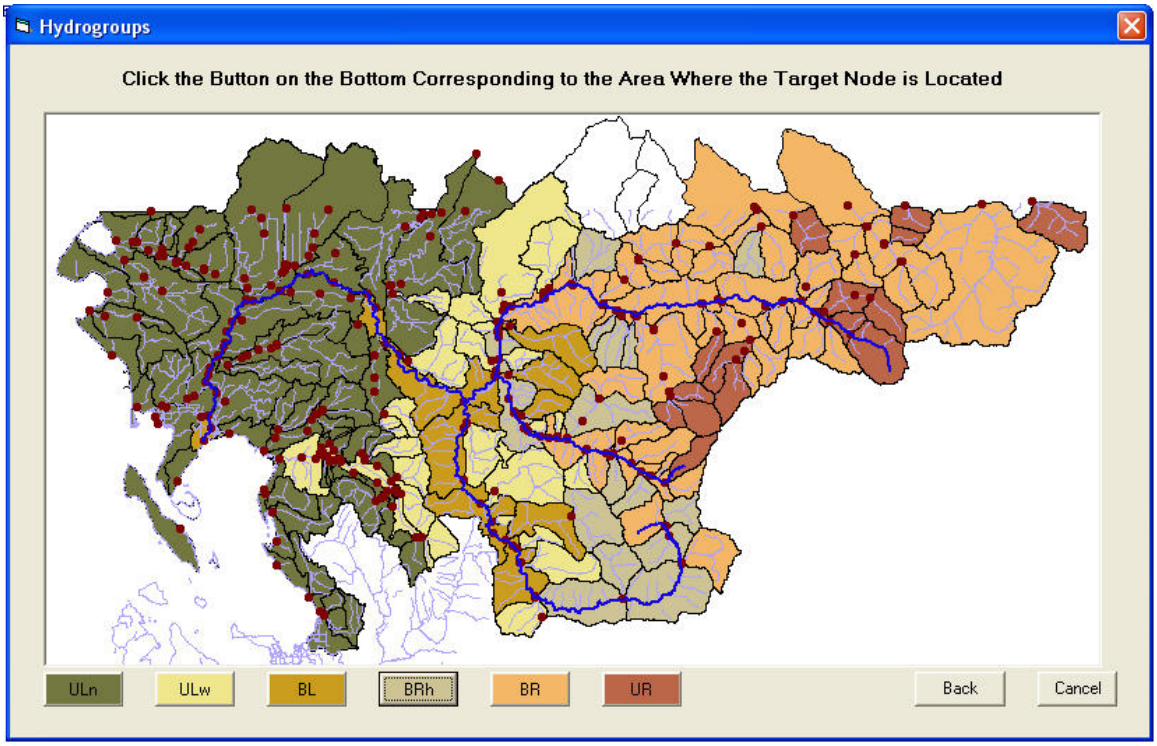



Figure 13. Nooksack watershed: Hydrogroups.

When the user clicks on one of the 6 buttons on the bottom, thus choosing the area where the target node is located, the next window will display that area with study sites and nodes belonging to that hydro group. Subclusters within each hydro group are shown in different colors (Figure 14). The user is encouraged to choose the source and target sites belonging to the same subcluster, but will be allowed to choose any source-target site combination provided both sites belong to the same hydro group. The user is prompted to enter the node number. The user can also choose to see a larger image by clicking “View Larger Image” button. Pressing Enter or clicking the “Close” button will close the new window.

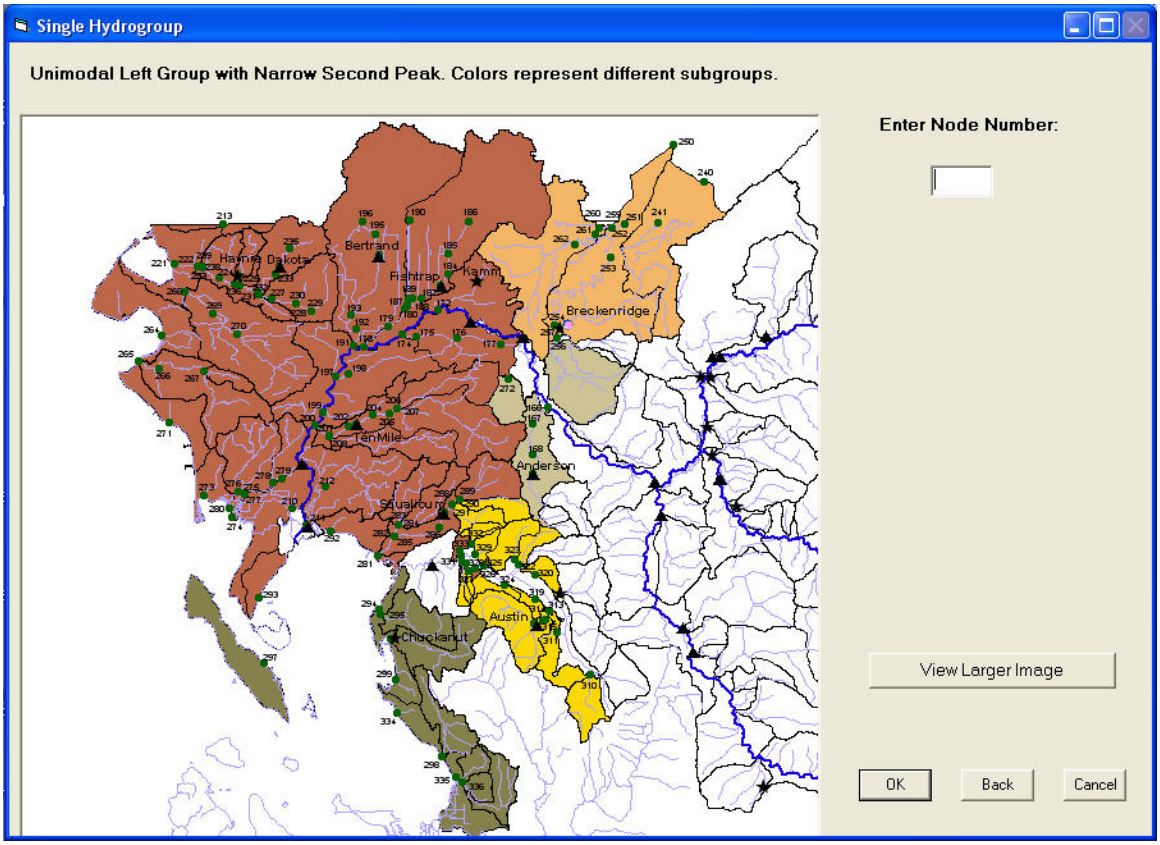



Figure 14. Example of Uln hydrogroup.

The next steps are the same as described in the "Enter the Node Number" Option. (Figures 9 to 12).

If extrapolating WUA curves, the user will be asked whether to output extrapolated values into a single .csv file or into the Habitat database (Figure 7) as described in "Extrapolate to All 300 nodes / Extrapolation of the WUA curves" section.

***Other Extrapolation Options***

If the user would like to extrapolate WUA curves to a number of nodes and use different source sites from those provided in the "Source-Target_337nodes.csv" file, the user can do it by either running the model for each node separately or by making corresponding changes in the 'Source_Site' column of the "Source-Target" file (replacing existing source sites with the new source sites).

When the program is finished running, the user gets the message: "Extrapolation is finished!"

# Habitat Prediction Program

## Introduction

The program predicts habitat values at small streams (mean flow < 130cfs) within the Nooksack watershed.

***In general, a stream where the proposed technique is expected to work well for all fish and life stages should not experience dramatic changes in substrate and/or cover availability from low to high flows. The more substrate/cover becomes available with increasing flows compared to low flow conditions the more the two approaches are expected to deviate.***

For more information on limitations of this approach see Appendix A. For complete description of the technique see Saraeva (2006; http://www.engineering.usu.edu/uwrl/inse/nooksack/ Saraeva/Dissertation.pdf ), chapter 3 (p.42)

## Program Setup

Folder “Pictures” has to be in the same folder as the program itself. The folder contains images of the western and eastern parts of the watershed that are displayed during the program run. The folder is provided with the program.

## Run Program

The user is given a choice to predict habitat values at a single node or a number of nodes with MAF of less than 130cfs (Figure 1). There are 252 of the 337 nodes with MAF of less than 130 cfs within the watershed.

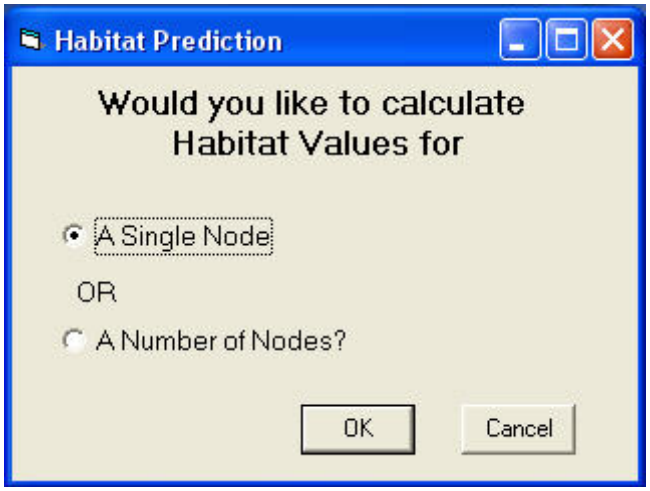



Figure 7. Habitat prediction choice.

The user is asked to open a comma delimited file with 7 columns: Node ID, Drainage ID, MAF (cfs), Drainage Area (sq.m), 10% exceedence flow (cfs), 90% exceedence flow (cfs), and Site Type. The file should have 338 rows: 337 nodes and a header. Flow_Area_10and90exceedences.csv file is provided with the program. Figure 2 shows an example of the file.

```
Node, DrainId,MAF_cfs, AreaTotal_sqm,10%Exceed_cfs,90%Exceed_cfs,Site_Type
1,113,3664.8,2.11E+09,6263.81894,1453.951797,Drainage Outlet
2,113,3652.2,2.06E+09,6225.322422,1450.537575,Intensive Site
3,46,3645.9,2.04E+09,6213.88047,1447.845185,Drainage Outlet
4,46,3645.7,2.04E+09,6213.831029,1447.832119,Water Withdrawal
5,46,3645.6,2.04E+09,6213.792183,1447.820465,Intensive Site
6,46,3645.5,2.04E+09,6213.767463,1447.814462,Water Withdrawal
7,46,3645.4,2.04E+09,6213.753337,1447.810577,Streamflow Gage
8,46,3619.1,1.95E+09,6179.861851,1441.978007,Drainage Outlet
9,46,3609.9,1.93E+09,6167.981997,1439.540588,Drainage Outlet
10,46,3609.9,1.93E+09,6167.918431,1439.538116,Water Withdrawal
11,46,3609.8,1.93E+09,6167.748921,1439.409571,Water Withdrawal
12,46,3597.6,1.90E+09,6142.774388,1435.555328,Drainage Outlet
13,46,3531.1,1.79E+09,6032.444306,1419.086333,Drainage Outlet
14,46,3475.9,1.69E+09,5924.678069,1409.137132,Drainage Outlet
15,46,3475.7,1.69E+09,5924.46265,1409.103583,MixingNode
```

Figure 8. Example of the input file.

```
Exported Fish Curves
H  10000  8  9 92  0 Chinook Spawn Stream
V  10000  0.00  0.00  0.50  0.00  1.00  0.90  1.75  1.00  2.25  1.00  4.00  0.00
V  10000  5.00  0.00 100.00  0.00
D  10000  0.00  0.00  0.50  0.00  1.20  1.00  1.50  1.00  3.00  1.00  3.50  0.50
D  10000  4.50  0.07  5.00  0.00 100.00  0.00
S  10000  0.00  0.00  0.10  1.00  0.20  1.00  0.30  1.00  0.40  1.00  0.50  1.00
S  10000  0.60  0.00  0.70  0.00  0.75  0.00  0.80  0.00  0.85  0.00  0.90  0.00
S  10000 11.90  0.00 31.70  0.00 31.80  0.24 31.90  0.27 32.50  0.00 32.70  0.00
S  10000 32.80  0.24 32.90  0.27 33.90  0.30 34.50  0.65 34.90  0.37 35.50  0.65
S  10000 35.90  0.37 36.50  0.65 36.90  0.37 37.50  0.40 37.90  0.32 38.50  0.00
S  10000 41.70  0.00 41.80  0.80 41.90  0.90 42.50  0.00 42.70  0.00 42.80  0.80
S  10000 42.90  0.90 43.50  0.65 43.90  0.93 44.90  1.00 46.90  1.00 47.50  0.75
S  10000 47.90  0.95 48.50  0.00 51.70  0.00 51.80  0.80 51.90  0.90 52.50  0.00
S  10000 52.70  0.00 52.80  0.80 52.90  0.90 53.50  0.65 53.90  0.93 54.50  1.00
S  10000 56.90  1.00 57.50  0.75 57.90  0.95 58.50  0.00 61.70  0.00 61.80  0.80
S  10000 61.90  0.90 62.50  0.00 62.70  0.00 62.80  0.80 62.90  0.90 63.50  0.65
S  10000 63.90  0.93 64.50  1.00 66.90  1.00 67.50  0.75 67.90  0.95 68.50  0.00
S  10000 71.70  0.00 71.80  0.40 71.90  0.45 72.50  0.00 72.70  0.00 72.80  0.40
S  10000 72.90  0.45 73.50  0.40 73.90  0.48 74.50  0.75 74.90  0.55 75.50  0.75
S  10000 75.90  0.55 76.50  0.75 76.90  0.55 77.90  0.50 78.50  0.25 78.90  0.00
S  10000 99.90  0.00100.00  0.00
```

Figure 9. First few records of the Master_WRIAHSC.txt

Next, the user is asked to open a text file (.txt) containing habitat values (Figure 3). Master_WRIA1HSC.txt is provided with the program.

If the user chooses to predict habitat at a number of nodes the program will ask for a comma delimited (.csv) file containing a list of nodes later in the run (Figure 5). The file provided with the program Nodes.csv contains a list of 253 nodes. To avoid selecting nodes with MAF higher than 130 cfs, any subset of the target nodes should be chosen from this file. If, however, there are some nodes in the file the user opens with MAF

higher than 130 cfs the user will get a message with an option to either skip the nodes with high MAF or quit the program (Figure 6).
The user is given a choice to output predicted habitat values into individual comma delimited file(s) or the "tblWUA_MethodC" table in the database used as an input in the Habitat Time Series Model (within DSS) – Figure 4. Detailed description of the table and the database in given in the Technical Documentation: The Habitat Time Series Model. HabitatTimeSeriesInput.mdb database is provided with the program.

If the user chooses to output habitat values into individual .csv files, separate for each node comma delimited files with predicted habitat values will be saved in the 'PredictedHabitat_Nodes' folder. If the folder already exists the user will be asked to either replace the folder or to enter a different folder name.

If the user chooses to output habitat values into the database, the user will be prompted to open the database.

If the user chooses to predict habitat at a single node, the program will display a window with the choice to either enter the node number or to see the map (Figure 7).

If the entered number is not a valid node number the user will be asked the reenter the node number.

If the entered node has a MAF of more than 130 cfs the user gets a message (Figure 8) and is asked to enter another node number.

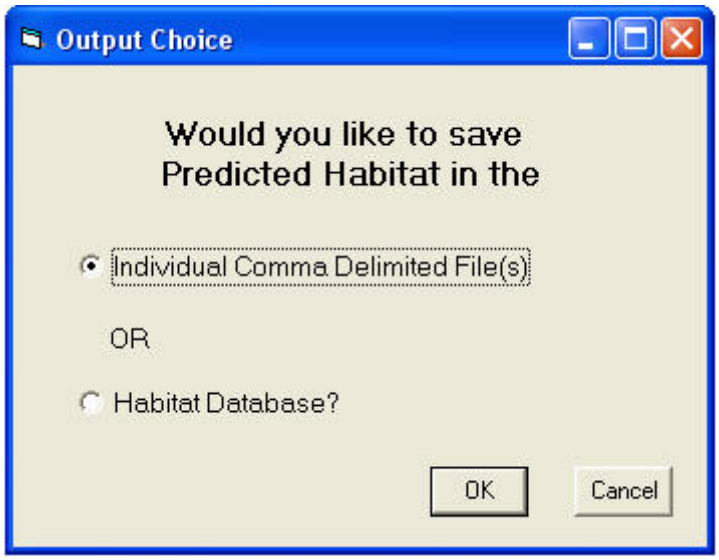



Figure 10. Predicted habitat output choice.

45
48
49
50
51
52
53
54
55
56
59
60
61
62
63
64

Figure 11. First few records of the Nodes.csv file.

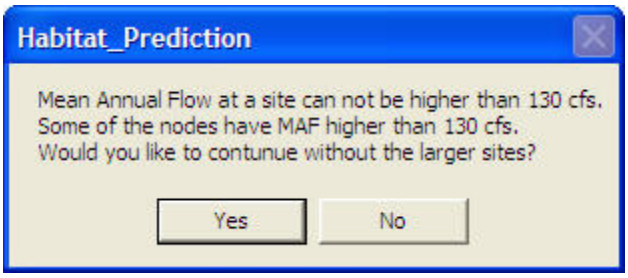

Habitat_Prediction
Mean Annual Flow at a site can not be higher than 130 cfs.
Some of the nodes have MAF higher than 130 cfs.
Would you like to contunue without the larger sites?
Yes
No

Figure 12. The message the user gets if there are some nodes in the node file with MAF > 130cfs.

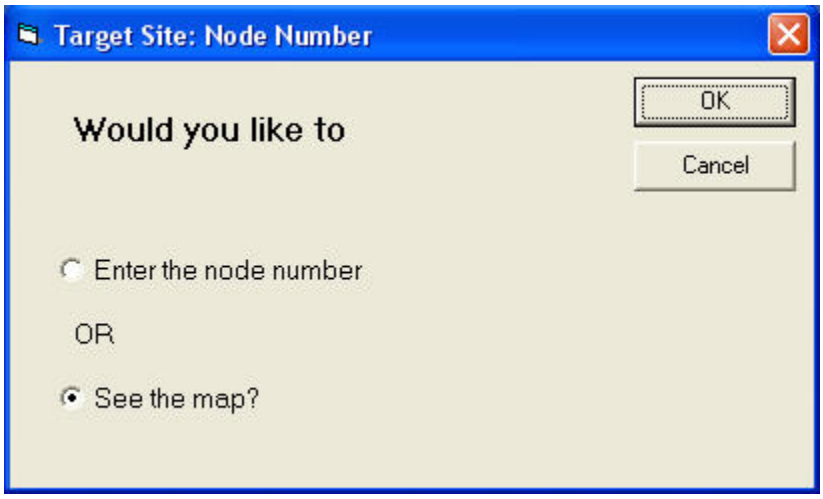



Figure 13. Entering the target site node number.

Figure 14. The message the user gets if the MAF at the target node is greater that 130 cfs.

If the user chooses to see the map the next window displays western or eastern part of the watershed (navigate between the western and eastern parts using 'Next>' and '<Previous' buttons) – Figure 9.

After the node number has been entered the user is given a choice to either use 10% and 90% exceedence flows as maximum and minimum flows (option "Yes") or manually enter the maximum and minimum flows for the calculations (option "No"; Figure 10). If option "Yes" is chosen the option of entering min and max flows (on the right part of the form) is not available.

Habitat values are then calculated for 20 flows in the range between the min and max flows.

After the program finished running the use gets a message: 'Calculations are finished!'

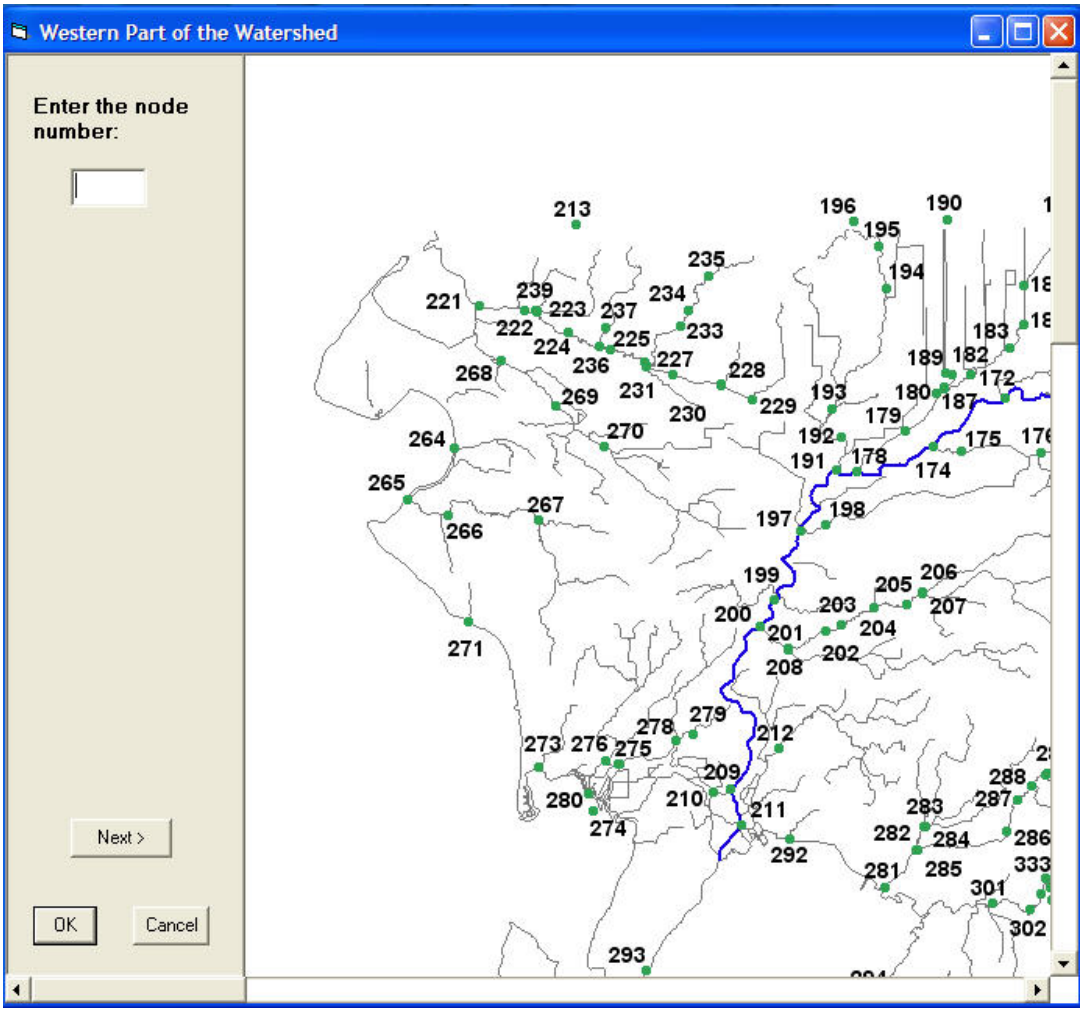



Figure 15. Maps of the node locations.

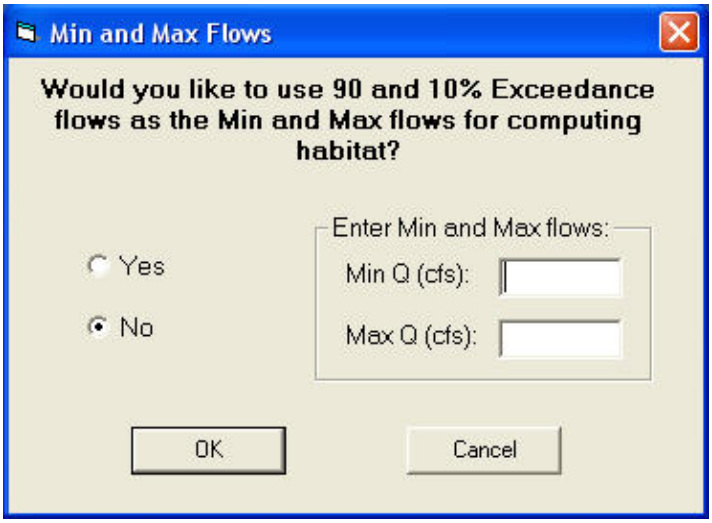

Min and Max Flows
Would you like to use 90 and 10% Exceedance flows as the Min and Max flows for computing habitat?
Yes
No
Enter Min and Max flows:
Min Q (cfs):
Max Q (cfs):
OK
Cancel

Figure 16. Entering min and max flows for the habitat calculation.

# Appendix C-1: Limitations of the Approach

The relationships between available habitat and discharge using the frequency based approach were compared to the WUA curves calculated using the two-dimensional habitat model. Results indicated that in most cases the two sets of habitat availability curves were very similar in shape (Figure C1-1) but differences were noted depending on the specific life stage being compared and attributed to the fundamental differences between frequency-based approach versus using the two-dimensional habitat model that incorporated substrate and cover for fry and juvenile life stages.

Available habitat for fry and juvenile life stages depends greatly on substrate conditions and availability of cover, which are not taken into consideration when calculating available habitat using the frequency equations. Figure C1-2 shows a comparison of the habitat curves for fry for Squalicum Creek. The frequency based predictions of available habitat for fry fit quite well at the lower flows; while deviating more at higher flows where substrate and cover become important factors contributing to estimated habitat using the two-dimensional habitat model.

The main channel usually serves as good habitat for these life stages at low to medium flows. When the flows increase further, velocities within the main channel become too high for fry and juvenile use; at the same time, shallow waters over gravel bars or vegetation outside of the main channel become available, which is highly suitable for these two life stages.

These results show that when substrate and/or cover represent an important component of habitat availability, greater caution and expert judgment should be exercised when interpreting the frequency-derived estimates of available habitat.

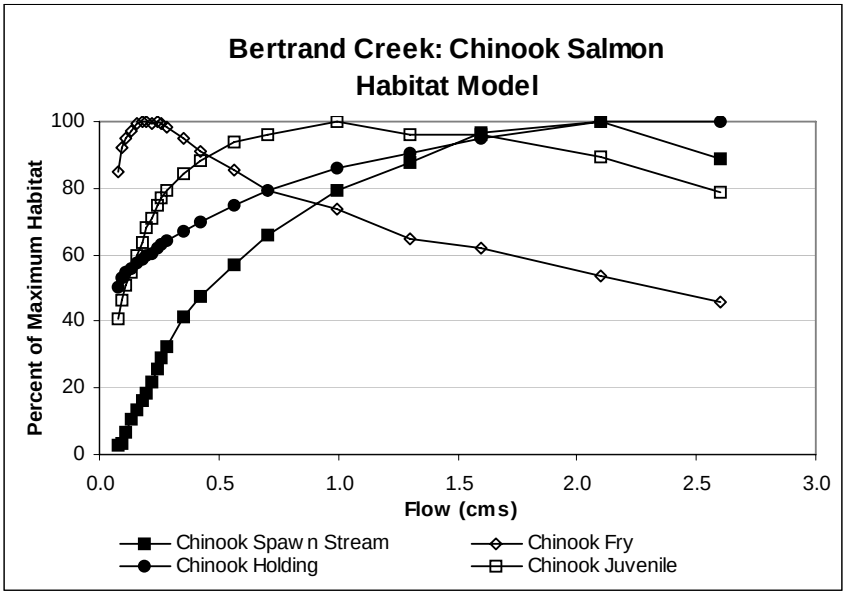
Figure C1-3 shows the RMSEs calculated by comparison of the two-dimensional habitat model and frequency based results for all small study sites for spawning, fry, juvenile, and adult life stages of selected fish species. Spawning and adult habitat relationships based on the frequency based approach are usually predicted with the lowest error (average RMSE 15.13% and 15.1%, respectively), whereas juvenile and fry habitat relationships derived from the frequency based approach have somewhat higher prediction errors (18.1% and 21.65%, respectively).

Substrate was a contributing factor in misrepresenting available habitat for rainbow and kokanee spawning at Whatcom creek at high flows (the two higher points – Figure C1-3). The shapes of available habitat versus discharge curves were similar to those shown in Figure C1-2 for both approaches.

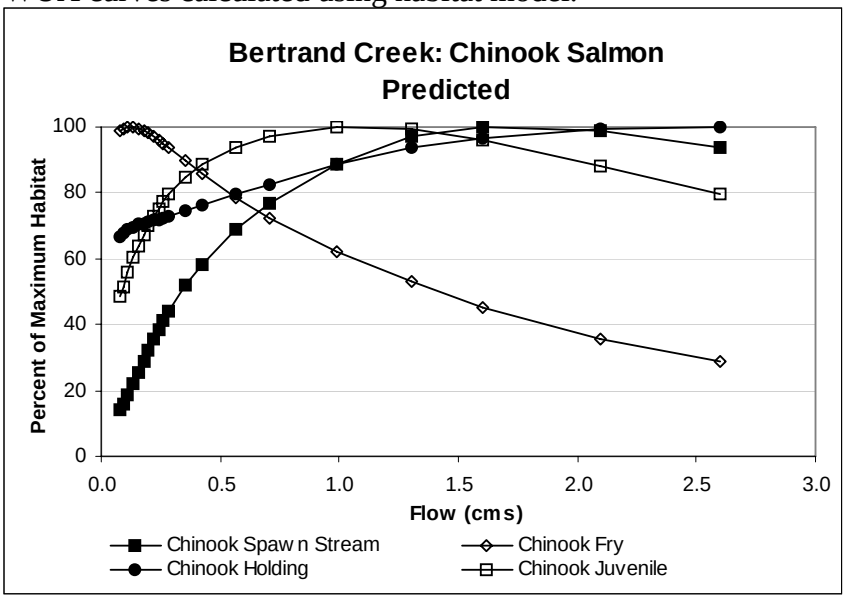
The Hutchinson Creek site was treated as an outlier throughout the analysis. The stream is characterized by a very distinctive main channel and adjacent relatively flat gravel

bars. The water begins to overtop the gravel bars in some places starting with relatively low flows, and then spreads out more with increasing flows, gradually.

The main channel usually serves as good habitat for these life stages at low to medium flows. When the flows increase further, velocities within the main channel become too high for fry and juvenile use; at the same time, shallow waters over gravel bars or vegetation outside of the main channel become available, which is highly suitable for these two life stages.




a. WUA curves calculated using habitat model.




b. WUA curves predicted using velocity and depth frequency distribution equations (frequency-based approach).

Figure C1--1. Comparison of habitat vs. discharge relationships for chinook salmon (Bertrand Creek).

These results show that when substrate and/or cover represent an important component of habitat availability, greater caution and expert judgment should be exercised when interpreting the frequency-derived estimates of available habitat.

Figure C1-3 shows the RMSEs calculated by comparison of the two-dimensional habitat model and frequency based results for all small study sites for spawning, fry, juvenile, and adult life stages of selected fish species. Spawning and adult habitat relationships based on the frequency based approach are usually predicted with the lowest error (average RMSE 15.13% and 15.1%, respectively), whereas juvenile and fry habitat relationships derived from the frequency based approach have somewhat higher prediction errors (18.1% and 21.65%, respectively).

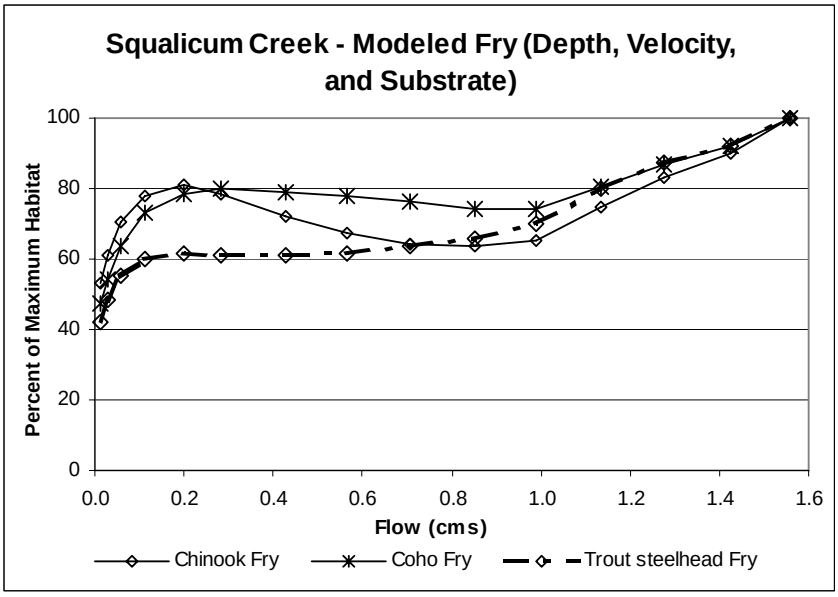
Substrate was a contributing factor in misrepresenting available habitat for rainbow and kokanee spawning at Whatcom creek at high flows (the two higher points – Figure C1-3). The shapes of available habitat versus discharge curves were similar to those shown in Figure C1-2 for both approaches.

The Hutchinson Creek site was treated as an outlier throughout the analysis. The stream is characterized by a very distinctive main channel and adjacent relatively flat gravel bars. The water begins to overtop the gravel bars in some places starting with relatively low flows, and then spreads out more with increasing flows, gradually providing more and more areas with depth and velocity suitable for all life stages of fish, so the available habitat for all target fish at this site increases with increasing flows. The frequency equations for depth work well for a regular shaped channel, but they are not able to reflect additional lower depths when the river overflows its active channel and inundates large flat areas. As a result, RMSEs for the majority of fish species and life stages for Hutchinson Creek were in the range between 35% and 45% (Figure C1-3). The hydraulic and habitat modeling were performed for flows ranging from 0.17 to 6.2 $m^3/s$, while the MAF at this site was 2.45 $m^3/s$. When a comparison of the habitat versus flow relationships are confined to flows ranging from 0.17 to 2.45 $m^3/s$ an average for the site RMSE was 15% (median RMSE – 18%).

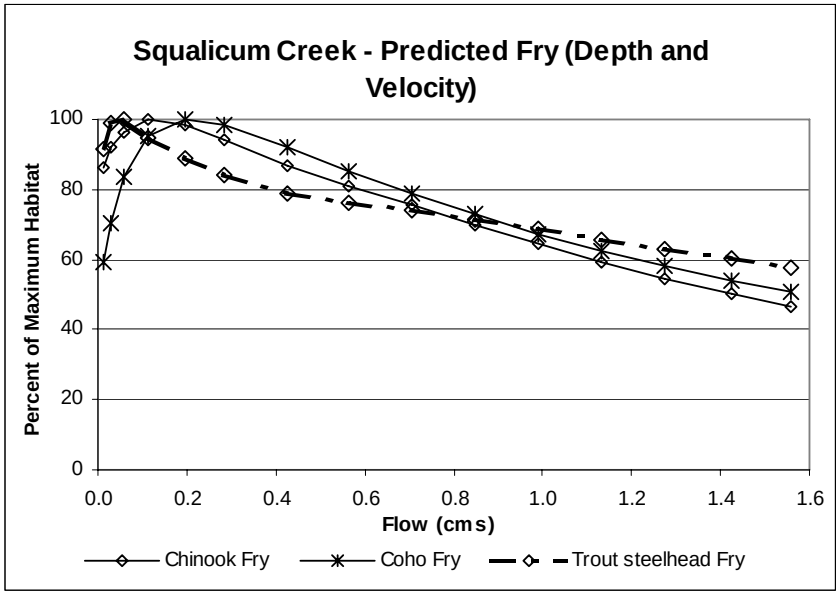
Tenmile Creek was another site that differed significantly from the rest of the intensively studied sites. The low flow channel of the stream is covered with sand (poor quality substrate/cover) while the margins are extensively covered with vegetation. When the flows increase the amount of high quality cover starts to increase significantly on both river banks (100% of additionally inundated area is high quality cover) highlighting the importance that substrate and cover can play in calculating available habitat. This leads to a misrepresentation of habitat curves for fry of all fish species (Figure C1-4), along with spawning and juvenile life stages of some species that are more sensitive to substrate when calculating habitat using the frequency equations.

In general, a stream where the proposed technique is expected to work well for all fish and life stages should not experience dramatic changes in substrate and/or cover

availability from low to high flows. The more substrate/cover becomes available with increasing flows compared to low flow conditions the more the two approaches are expected to deviate. Tenmile Creek is an example of an extreme case: high quality cover increased from none to about 40% of the entire wetted area as the flow increased from the 90% to about the 10% exceedence flow, and all of the additional area represented favorable substrate/cover for target life stages (fry, juvenile, spawning).




a. WUA curves calculated using habitat model.




b. WUA curves calculated using velocity and depth frequency distribution equations (frequency-based approach).

Figure C1-2. Comparison of habitat vs. discharge relationships for fry life stage for Squalicum Creek.

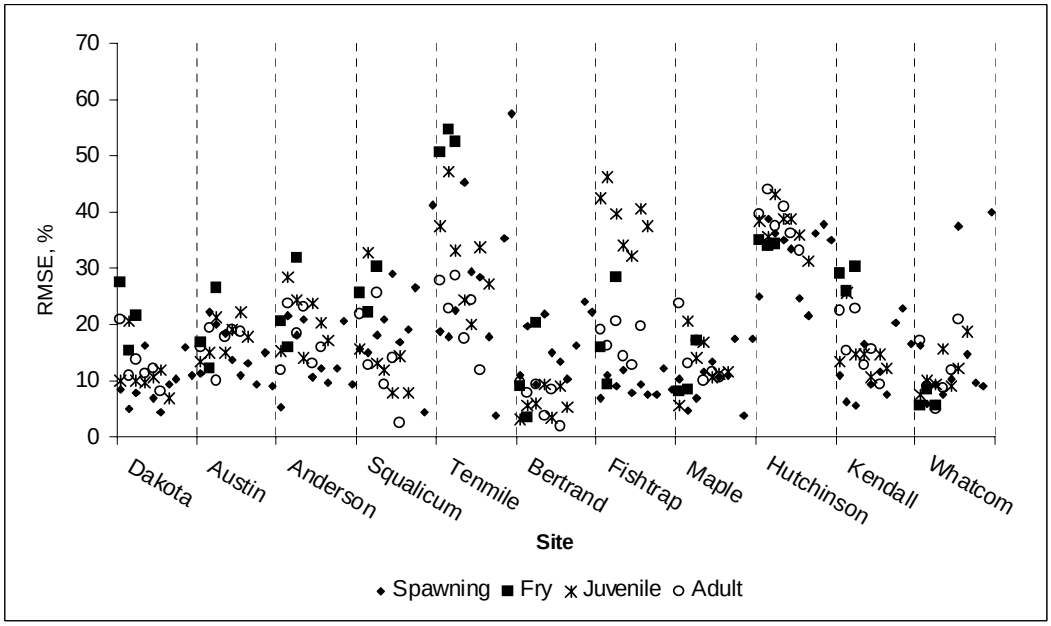


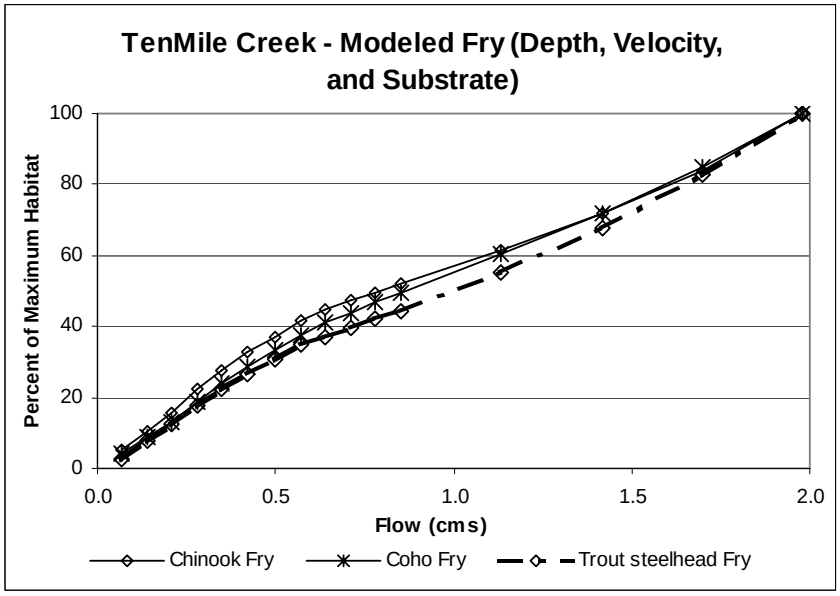

Figure C1-3. RMSEs for WUA curves (habitat model) and HV curves (frequency equations) for small study sites.

Substrate within Squalicum Creek included some cobble (good quality cover) at low flow, and as the flow increased to about the 10% exceedence flow almost all of the additional wetted area represented high quality substrate/cover for spawning, fry, and juvenile. In this instance, the frequency based approach gave a good approximation of available habitat for fry at flows below 0.7 $m^3/s$ (Figure CA2) where no significant

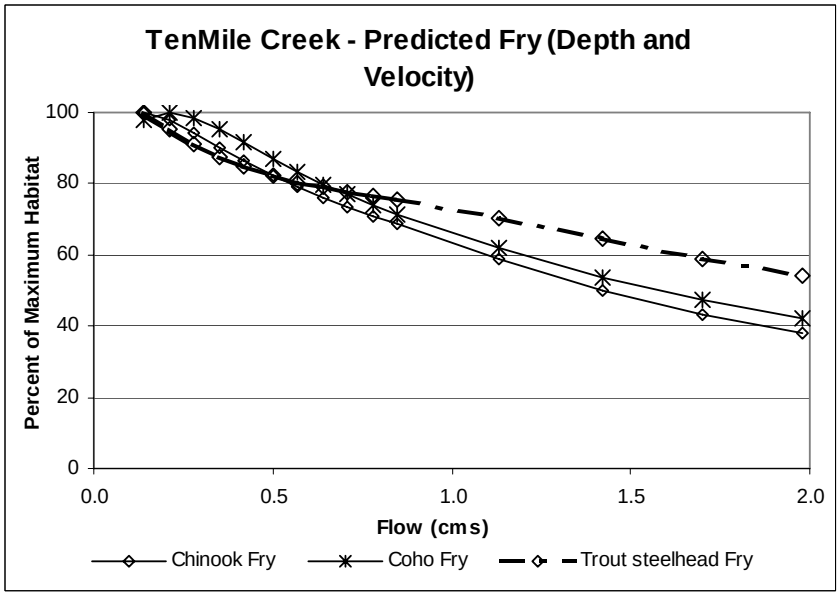
changes in substrate/cover conditions occurred, and misrepresented available habitat at higher flows where significantly more favorable conditions for fry substrate and cover became available. The most sensitive life stages to preferred substrate and cover availability used in our analyses are listed in Table C1-1 (RMSE higher than 10%).

Another factor affecting the results is the shape of suitability curves. As a rule, life stages that are characterized by very narrow ranges of optimum values on the suitability curves of at least one parameter (velocity or depth) but especially both parameters are expected to have higher prediction errors. Table C1-2 lists life stages that have higher prediction errors due to the shape of suitability index curves (RMSE above 10%).

Table C1-3 lists target fish and life stages with corresponding RMSEs averaged between small study sites. Since Hutchinson Creek had a unique shape of the riverbed and Tenmile Creek was characterized by significantly different substrate/cover conditions, they were omitted from these results. Most of the frequency based curves were predicted within a RMSE of 15% overall, and only two life stage curves were predicted with a RMSE of more than 20%.




a. WUA calculated using habitat model.




b. WUA curves calculated using velocity and depth frequency distribution equations (frequency-based approach).

Figure C1-4. Comparison of habitat vs. discharge relationships for fry life stage for Tenmile Creek.

Table C1-1. Life stages of target fish species most sensitive to substrate and cover conditions. RMSEs were estimated for WUA curves calculated using habitat model when substrate and cover were taken into consideration and WUA curves calculated using habitat model without inclusion of substrate and cover. Hutchinson Creek was excluded from the analysis.

| Fish Species/ Life Stage | RMSE, % |
|---|---|
| Kokanee Spawning | 14.04 |
| Coho Juvenile | 12.82 |
| Trout steelhead Fry | 12.63 |
| Chinook Fry | 12.04 |
| Coho Fry | 11.73 |
| Pink Spawning | 10.80 |
| Cutthroat Spawning | 10.43 |
| Native char Spawnin | 10.09 |
| Rainbow Spawning | 9.71 |

Table C1-2. Life stages of target fish species that have narrow ranges of optimum parameters. RMSEs were estimated for WUA curves calculated using habitat model without inclusion of substrate and cover and HV curves predicted using frequency equations. Hutchinson Creek was excluded from the analysis.

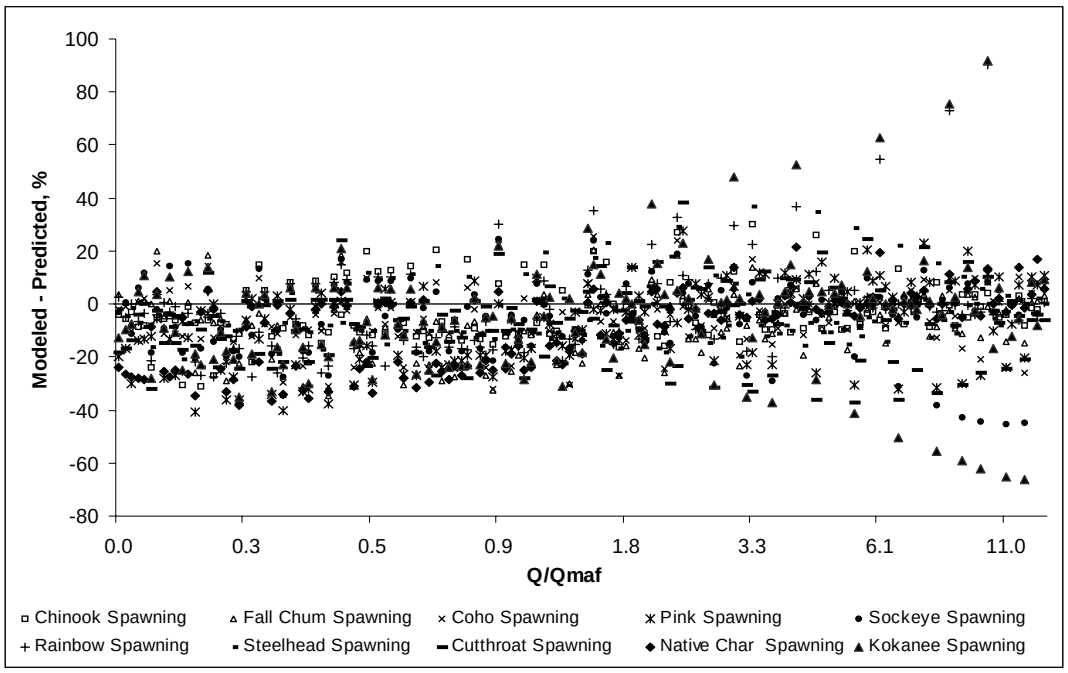
| Fish/Life Stage | RMSE, % | Fish/Life Stage | RMSE, % |
|---|---|---|---|
| Sockeye Spawning | 10.23 | Steelhead Juvenile | 12.94 |
| Chinook Juvenile | 10.28 | Rainbow Adult | 13.44 |
| Coho Spawning | 10.74 | Rainbow Juvenile | 13.60 |
| Cutthroat Adult | 11.14 | Cutthroat Spawning | 13.99 |
| Cutthroat Juvenile | 11.24 | Coho Fry | 14.40 |
| Rainbow Spawning | 11.35 | Rainbow Winter | 14.74 |
| Mountain Whitefish Adult | 11.54 | Chinook Fry | 15.44 |
| Mountain Whitefish Juvenile | 12.30 | Pink Spawning | 17.55 |
| Native Char Adult | 12.38 | Coho Juvenile | 17.62 |
| Native Char Juvenile | 12.69 | Chinook Holding | 19.31 |
| Fall Chum Spawning | 12.70 | Steelhead Fry | 19.76 |
| Kokanee Spawning | 12.86 | Native Char Spawnin | 20.19 |

Table C1- 3. RMSEs calculated for WUA curves (habitat model) and HV curves (proposed technique) for small sites within WRIA1. Hutchinson Creek and Tenmile Creek were excluded from the analysis.

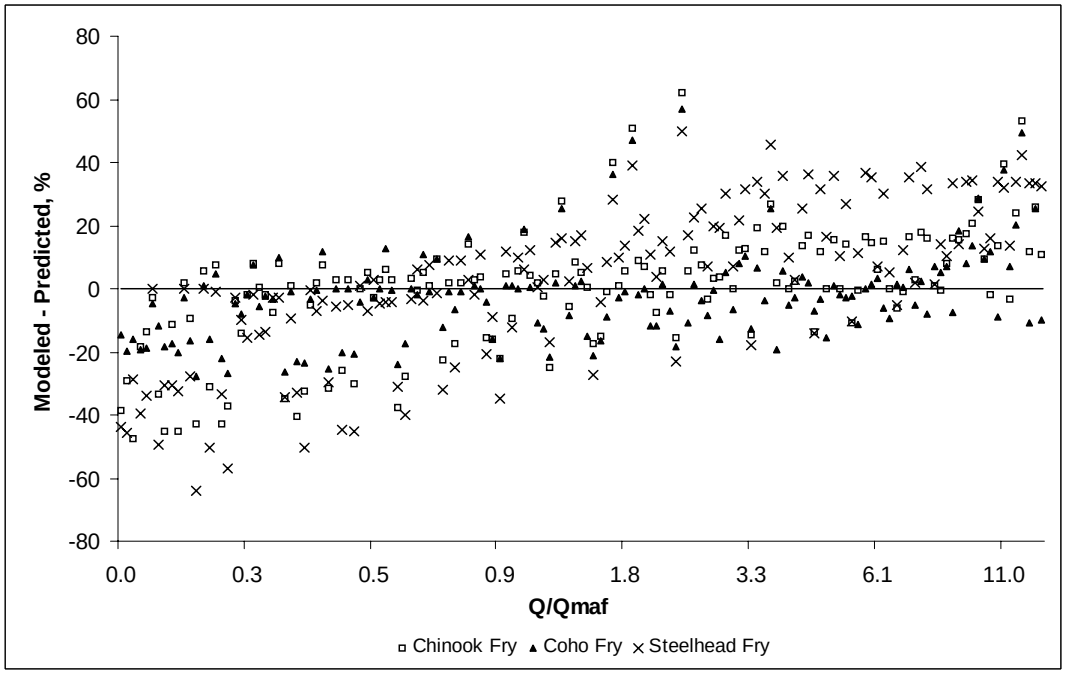
| Fish/Life Stage | RMSE, % | Fish/Life Stage | RMSE, % |
|---|---|---|---|
| Chinook Spawn Stream | 10.65 | Rainbow Adult | 14.31 |
| Steelhead Spawning | 11.40 | Cutthroat Spawning | 14.41 |
| Coho Spawning | 11.58 | Mountain Whitefish Juvenile | 14.99 |
| Mountain Whitefish Adult | 12.01 | Steelhead Juvenile | 15.70 |
| Cutthroat Adult | 12.36 | Rainbow Winter | 15.84 |
| Fall Chum Spawning | 12.38 | Pink Spawning | 16.23 |
| Sockeye Spawning | 12.92 | Rainbow Juvenile | 16.99 |
| Native Char Adult | 13.19 | Native Char Juvenile | 17.40 |
| Coho Fry | 13.44 | Chinook Fry | 17.59 |
| Rainbow Spawning | 14.07 | Chinook Holding | 18.10 |
| Chinook Juvenile | 14.07 | Kokanee Spawning | 19.47 |
| Cutthroat Juvenile | 14.09 | Coho Juvenile | 22.78 |
| Native Char  Spawning | 14.23 | Steelhead Fry | 23.59 |

When a comparison was made between WUA curves calculated using 2d hydraulic and habitat models and WUA curves calculated using 1d hydraulic and habitat models for the same study sites, average RMSE was 10.5% (median RMSE – 8.1%); when WUA curves calculated using 2d models were compared to Habitat Values calculated using frequency equations an average RMSE was 15.2% (median RMSE – 13.4%).

Figure C1-5 shows prediction error residuals between WUA curves derived using the habitat model and calculated using the frequency-based approach for each modeled flow normalized by the MAF at the study site (Tenmile and Hutchinson creeks were excluded from this analysis). Available habitat for spawning was over predicted at low flows and had low prediction errors at high flows for all fish species except kokanee salmon and rainbow trout; for these two fish species prediction errors were extremely high at Whatcom and Squalicum Creeks at higher flows due to reasons discussed above. Available habitat for the fry life stages were on average over predicted at low flows and under predicted at high flows. Habitat for juvenile and adult life stages was on average over predicted at all flows. Steelhead fry, coho juvenile, and rainbow winter were the life stages predicted with the highest error (also shown in Table C1-3).

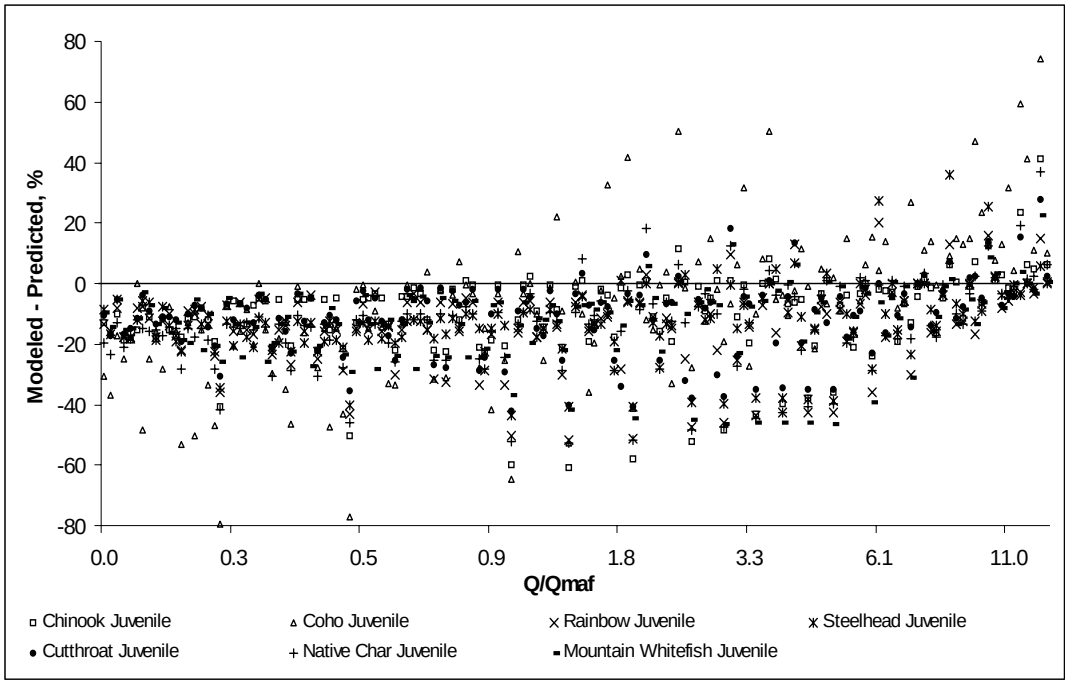



a. Prediction errors for the Spawning life stage of the select fish species.

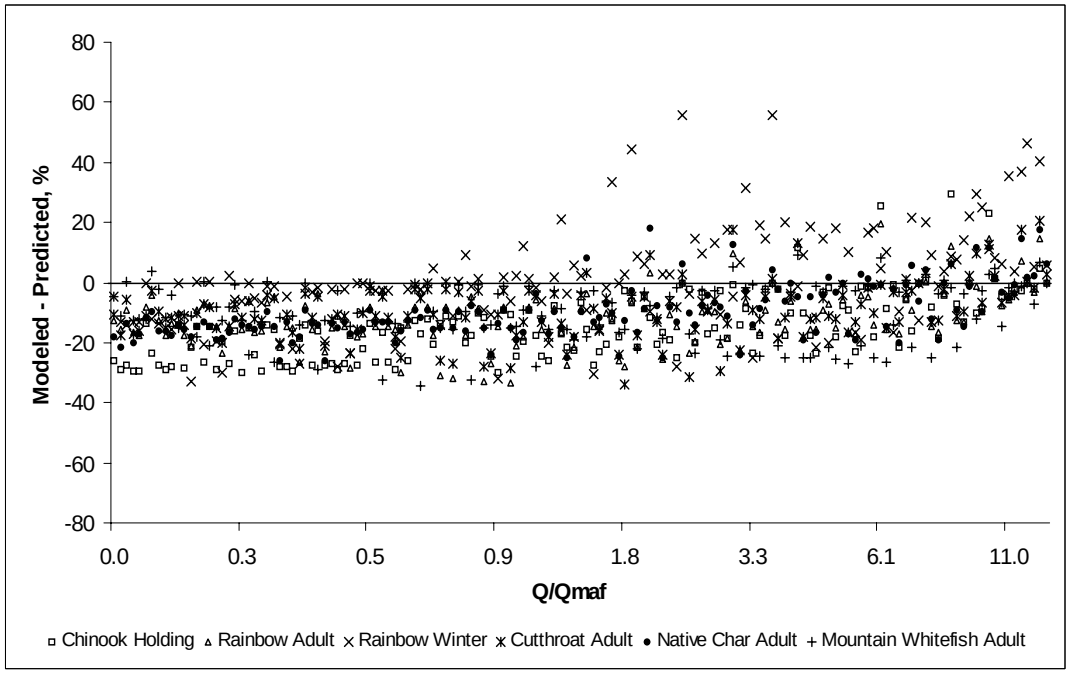



b. Prediction errors for the Fry life stage of the select fish species.

Figure C1-5. Prediction errors between habitat vs. discharge relationships derived using the habitat model and calculated using the frequency-based approach. (cont.)




c. Prediction errors for the Juvenile life stage of the select fish species.




d. Prediction errors for the Adult life stage of the select fish species.

Figure C1-5 Continued.